Rocket Propulsion Analysis

Version 2.3

User Manual

Cologne, Germany – 2017
www.rocket-propulsion.com

Table of Contents

Introduction

RPA is an acronym for Rocket Propulsion Analysis.

RPA is an easy-to-use multi-platform tool for the performance prediction of rocket engines. It features an intuitive graphical user interface with convenient grouping the input parameters and analysis results. RPA utilizes an expandable chemical species library based on NASA Glenn thermodynamic database, that includes data for numerous fuels and oxidizers, such as liquid hydrogen and oxygen, kerosene, hydrogen peroxide, MMH, and many others. With embedded species editor, the users may also easily define new propellant components, or import components from *PROPEP* or *CEA2* species databases.

By providing a few engine parameters such as combustion chamber pressure, used propellant components, and nozzle parameters, the program obtains chemical equilibrium composition of combustion products, determines its thermodynamic properties, and predicts the theoretical rocket performance. The results of calculation can also be used to design combustion chambers, gas generators and preburners of the liquid propellant rocket engines.

The calculation method is based on robust, proven and industry-accepted Gibbs free energy minimization approach to obtain the combustion composition, analysis of nozzle flows with shifting and frozen chemical equilibrium, and calculation of engine performance for a finite- and infinite-area combustion chambers.

RPA is written in C++ programming language using following libraries: Qt, Qwt, libconfig++.

The program was written by Alexander Ponomarenko. You can contact him by sending an email to: contact@propulsion-analysis.com

RPA editions

You can download three different versions of RPA from http://www.propulsion-analysis.com/downloads.htm: freeware Lite Edition, commercial Standard Edition v.1.x and commercial Standard Edition v.2.x.

System requirements and installation procedure are the same for both editions.

If you downloaded and used an evaluation copy of RPA Standard Edition with free 15-day trial period, you may purchase a license and get your personalized product key which converts this evaluation version of the product to a fully licensed version.

System requirements

Microsoft Windows

Operating Systems:

- Windows XP, Windows Vista, Windows 7, Windows 8, and Windows 10 (32-bit or 64-bit Edition)

Any computer that runs with mentioned operating systems.

Apple Mac

- Mac OS X 10.5 or later
- Macintosh computer with an Intel x86-64 processor

Linux

RPA will not run without the following libraries:

- Glib 2.12 or higher
- X.Org 1.0 or higher

Installation on Microsoft Windows

RPA for Microsoft Windows is distributed in installation and ZIP packages both for x86 and x86-64 architectures.

RPA for Windows depends on Qt and MS VC++ 2010 run-time libraries. If your computer does not have it installed, please choose the package that includes all required components.

If you downloaded installation package

- Run installation executable file and follow the instructions the installer provides
- When done with the installation, you can delete the installer file to recover disk space
- The installer will create shortcuts for RPA executable on desktop and start menu, which you can use to start the application
- To uninstall the application run `Uninstall.exe` from the RPA installation directory

Note that in order to install the software from installation executable file you must have administrator rights. If you don't have administrative rights, you can still install the program from ZIP package.

If you downloaded ZIP package

- Extract files from the ZIP package into selected directory
- Start the program, executing the command `RPA.exe`
- To uninstall the application delete the RPA installation directory

Installation on Apple Mac OS X

RPA for Apple Mac OS X is distributed in zip package, containing binary of RPA for Intel x86-64 architecture as well as all required third-party libraries.

To install the program

- Extract files from the archived package into selected directory
- Start the program, executing the shell script `RPA.command`
- To uninstall the application delete the RPA installation directory

Installation on Linux

RPA for Linux is distributed in tar.gz packages both for x86 and x86-64 architectures.

RPA for Linux depends on Qt libraries. If your computer does not have it installed, please choose the package that includes all required components.

To install the program

- Extract files from the archived package into selected directory
- Start the program, executing the shell script `RPA.exe`
- To uninstall the application delete the RPA installation directory

Running RPA

Graphical User Interface

You start RPA either by clicking on an icon (on Desktop or in file browser), or typing `RPA.exe` on a command line.

Although command line arguments are not required when starting RPA, the available arguments are shown below:

Option	Value	Description
`-t` or `--thermo`	`FILE`	Thermodynamics database. Default is resources/thermo.inp
`-ut` or `--usr_thermo`	`FILE`	User-defined thermodynamics database. Default is resources/usr_thermo.inp
`-p` or `--properties`	`FILE`	Properties database. Default is resources/properties.inp

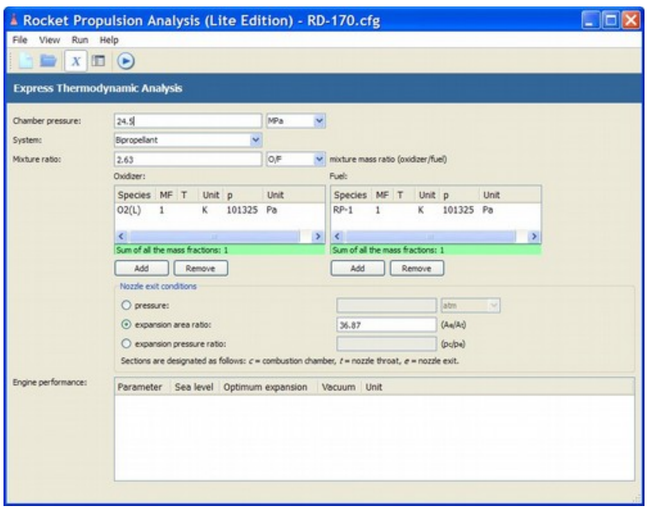
Option	Value	Description
`-up` or `--usr_properties`	`FILE`	User-defined properties database. Default is resources/properties.inp
`-i` or `--input`	`FILE`	Problem configuration file that has to be loaded. Default is last opened file.

Command line arguments must be in the command line that you use to start RPA.

See chapter Thermodynamic Database Editor for more information about database types.

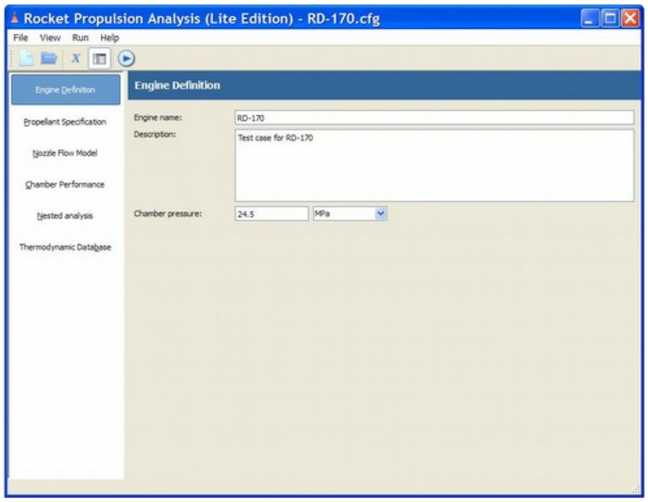
After starting the program, the RPA main window will appear. The main windows features menu bar, toolbar and working area, that can be used in two views: Express Analysis and Extended Analysis.

Express Analysis view is intended to keep the subset of input parameters and results on the same screen, and can be useful for quick analysis of the rocket engines, when theoretical performance is the only result that should be considered.

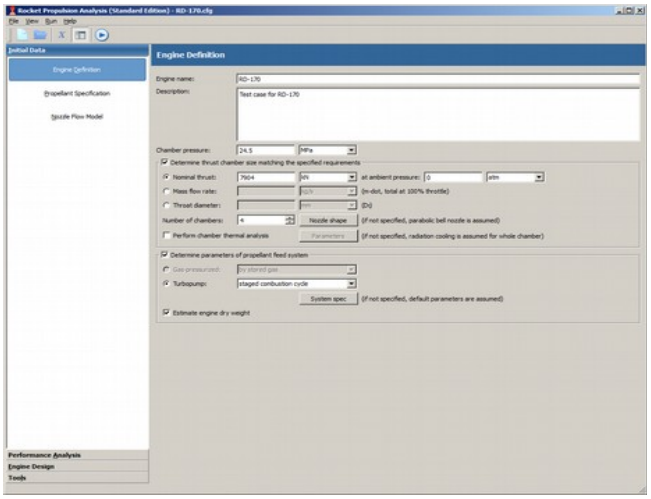


Express Analysis view

Extended Analysis view consists of several screens which conveniently group the input parameters and results. The desired screen can be activated by mouse click on corresponding button on the list at the left side of the main window. You can enlarge or narrow the list while the screens will be narrowed or enlarged, dragging the vertical bar between the list and the screens right or left.



Extended Analysis view (Lite Edition)



Extended Analysis view (Standard Edition)

In RPA Standard Edition, there are 4 different lists grouped in the following folders:

- **Initial Data** containing items Engine Definition, Propellant Specification, and Nozzle Flow Model,
- **Performance Analysis** containing items Chamber Performance, Nested Analysis, and Propwllant Analysis,
- **Engine Design** containing items Chamber Geometry, Thermal Analysis, and Propellant Feed System
- and **Tools** containing the item Thermodynamic Database.

Command-line utility

You start command-line utility by typing `rpac.exe` on a command line.

The available command-line arguments are shown below:

Option	Value	Description
`-t` or `--thermo`	`FILE`	Thermodynamics database. Default is resources/thermo.inp
`-ut` or `--usr_thermo`	`FILE`	User-defined thermodynamics database. Default is resources/usr_thermo.inp
`-p` or `--properties`	`FILE`	Properties database. Default is resources/properties.inp
`-up` or `--usr_properties`	`FILE`	User-defined properties database. Default is resources/properties.inp
`-i` or `--input`	`FILE`	Problem configuration file that has to be loaded. Default is last opened file.
`-o` or `--output`	`FILE_NAME_PREFIX`	Output file name prefix without extension. Default is "info".
`-opt` or `--optimize`		Find optimum propellant mixture ratio, bypassing the one

Option	Value	Description
		defined in input configuration file.
`-bau` or `--bau_units`		Print out results using britisj-american units.
`-pr` or `--performance`		Print out performance table
`-pt` or `--points`	`NUMBER`	Number of lines in altitude performance table

Upon completion, the command-line utility prints out the results in console window and writes it into the log file.

Scripting utility

Scripting utility is a tool that can be used to execute user's own problems.

Scripting utility can be started in either an interactive mode or a batch mode.

You start scripting utility by typing `rpas.exe` on a command line. The available command-line arguments are shown below:

The available command-line arguments are shown below:

Option	Value	Description
`-t` or `--thermo`	`FILE`	Thermodynamics database. Default is resources/thermo.inp
`-ut` or `--usr_thermo`	`FILE`	User-defined thermodynamics database. Default is resources/usr_thermo.inp
`-p` or `--properties`	`FILE`	Properties database. Default is resources/properties.inp
`-up` or `--usr_properties`	`FILE`	User-defined properties database. Default is resources/properties.inp
`-i` or `--input`	`FILE`	Script file.
`-o` or `--output`	`FILE_NAME_PREFIX`	Output file name prefix without extension. Default is "info".

Upon start up, scripting utility prints out the prompt `rpa>`, inviting you to type any valid command.

Type "exit" to stop the interactive interpreter. See Scripting Built-In Commands and Scripting API Reference to get more information about available commands.

To start scripting utility in the batch mode, specify the name of the script you want to execute as a command-line argument:

```
rpa.exe -i some_script.js
```

After completion, the scripting utility prints out the results in console window and writes it into the log file.

Configuration Files

The analysis problem input data is stored in the configuration file with extension `.cfg`. This is a specially formatted ASCII file, that can be viewed/edited in any ASCII text editor.

To start new analysis problem, create configuration file by clicking **File**, and then **New** in menu bar, or clicking icon **New** on the toolbar.

To continue with old analysis problem, load existing configuration file by clicking **File**, and then **Open** in menu bar, or clicking icon **Open** on the toolbar, and select the file to be opened. If you already have opened a configuration file, or have created new one, you will be asked to confirm the opening another configuration file.

To save the current analysis problem into the file, click **File**, and then **Save** in the menu bar. If the current analysis problem is new, you will be asked to specify the file name, or to select the existing file to be written. Otherwise the data will be saved into the same source file, that has been opened before.

You also can write the current analysis problem into another file, clicking **File**, and then **Save As...** in the menu bar.

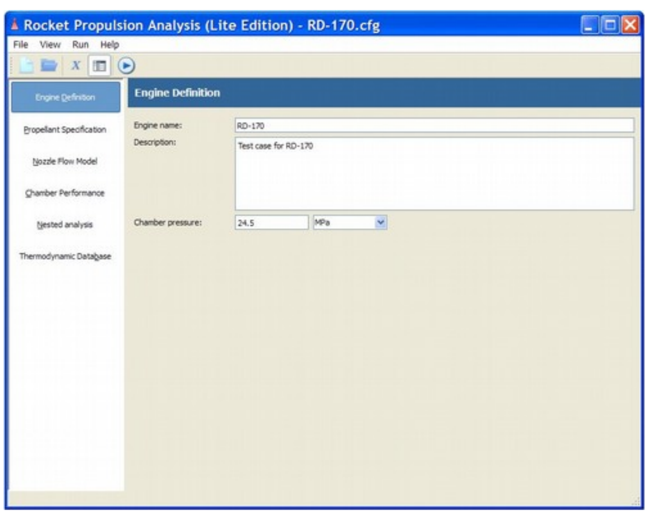
The last 10 used files are shown at the bottom of menu **File** in menu bar.

The last used configuration file is automatically opened at program start up.

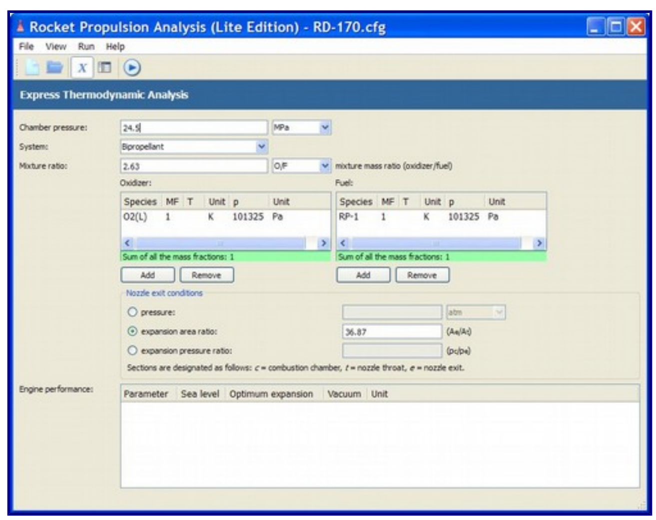
The program is shipped with a few example configuration files, located in directory examples.

Engine Definition

In the **RPA Lite Edition**, Engine Definition screen is used to define the engine name, the description and the combustion chamber pressure.



Engine Definition screen (Lite Edition)



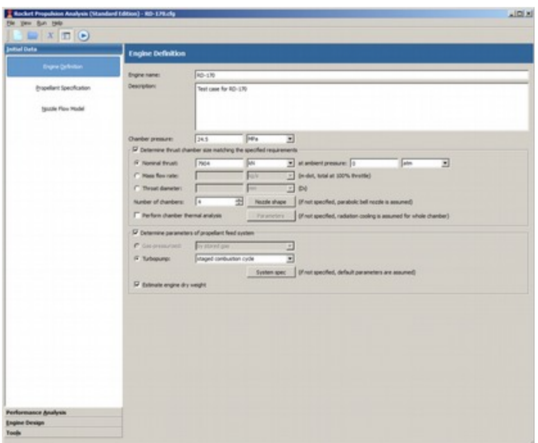
Engine Definition in Express Analysis view

Engine name and description are optional parameters, whereas the combustion chamber pressure is obligatory parameter. Note that engine name is used in results print out to identify the problem.

The pressure is an absolute pressure and can be entered using one of the following units: MPa, atm, kg/sm^2, bar, psi, Pa.

In the Express Analysis view the only parameter that is visible and can be changed is combustion chamber pressure.

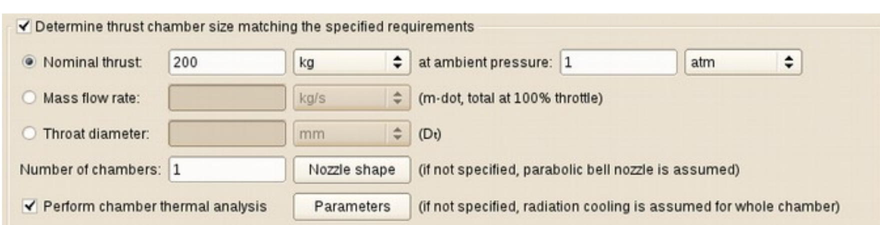
In the **RPA Standard Edition**, Engine Definition screen is also used to define the parameters for the combustion chamber and nozzle sizing, for switching on the flag for performing the thrust chamber thermal analysis, and to define the type of the engine cycle.



Engine Definition screen (Standard Edition)

The program can estimate the size of the combustion chamber and nozzle matching one of the following requirements:

- Nominal thrust at the certain ambient pressure
- Nominal mass flow rate
- Throat diameter



Chamber and nozzle sizing parameters

If specified number of chambers is greater than 1, the given thrust is a total engine thrust, and the given mass flow rate is a total mass flow rate.

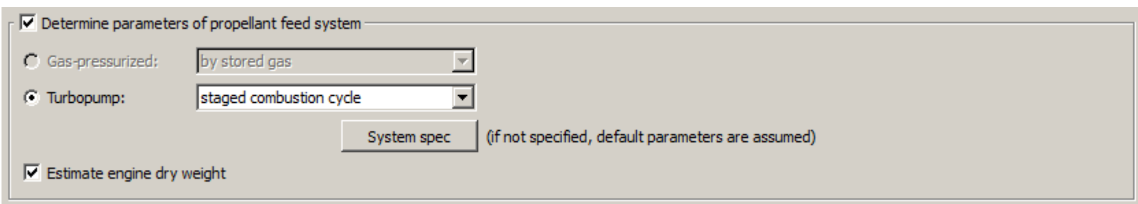
If ambient pressure is not specified, it is assumed that the engine nominally operates in vacuum condition.

The user can specify the nozzle shape on the screen Nozzle Shape and Efficiencies, and sizing parameters on the screens Nozzle Conditions and Chamber Geometry.

If flag “Perform chamber thermal analysis” is switched on, the program will execute the thermal analysis. The user can specify additional heat transfer and chamber cooling parameters on the screens Heat Transfer Parameters and Thrust Chamber Cooling.

The program can perform engine cycle analysis, obtaining parameters of liquid propellant feed system and estimating engine weight for one of the following engine cycles:

- Gas generator cycle
- Staged combustion cycle
- Full flow staged combustion cycle



Engine cycle type

Additional parameters for cycle analysis can be specified on the screen Propellant Feed System.

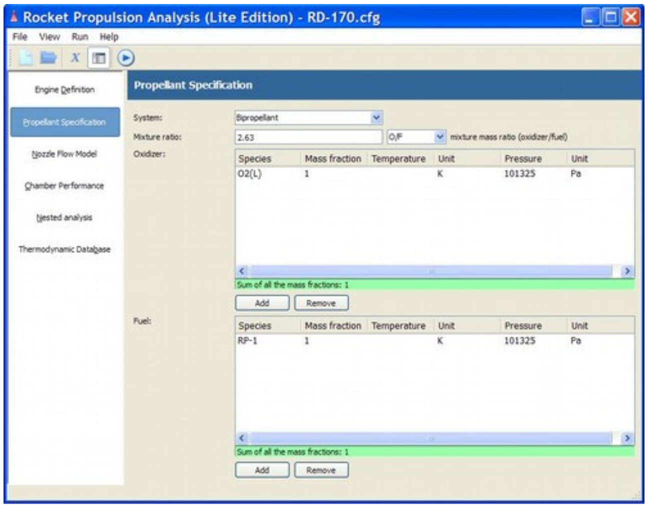
All parameters can be entered using either SI or American Customary units:

- thrust: `kN`, `kg`, `lbf`, `N`
- mass flow rate: `kg/s`, `lbm/s`
- throat diameter: `mm`, `in`, `m`, `ft`

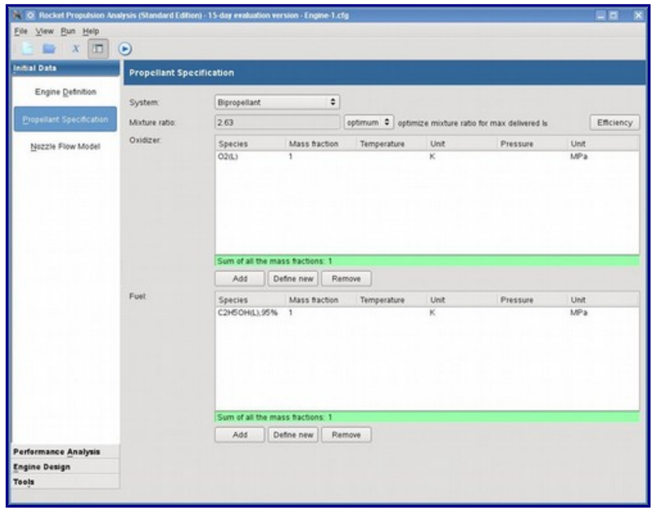
Propellant Specification

Propellant Specification Screen

Propellant Specification screen is intended to specify used propellant component/s.

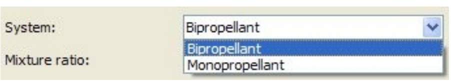


Propellant Specification screen (Lite Edition)



Propellant Specification screen (Standard Edition)

You can choose between bipropellant and monopropellant propulsion systems, selecting the corresponding item in the list box at the top of the screen:



Note: although you have the choice between bipropellant and monopropellant propulsion systems only, *there is a possibility to specify three (or more) propellant components*.

See section **How to...** (http://www.propulsion-analysis.com/howto/index.htm) on RPA web site for further details.

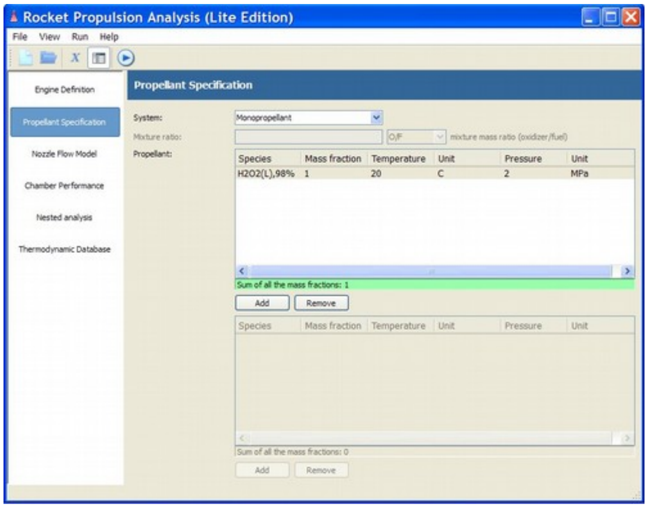
For bipropellant systems, the lists for both Oxidizer and Fuel are enabled (see figure *"Propellant Specification screen"* above), as well as the fields for specifying a mixture ratio.

The mixture ratio can be specified either as an O/F ratio (ratio of "oxidizer flow rate" to "fuel flow rate"), or as an oxidizer excess coefficient, given as ratio of desired O/F to stoichiometric O/F.

You can also select an item "optimum":

In this case the mixture ratio will be optimized for getting maximum specific impulse under given conditions. The found optimum O/F ratio will be displayed on the screen Chamber Performance.

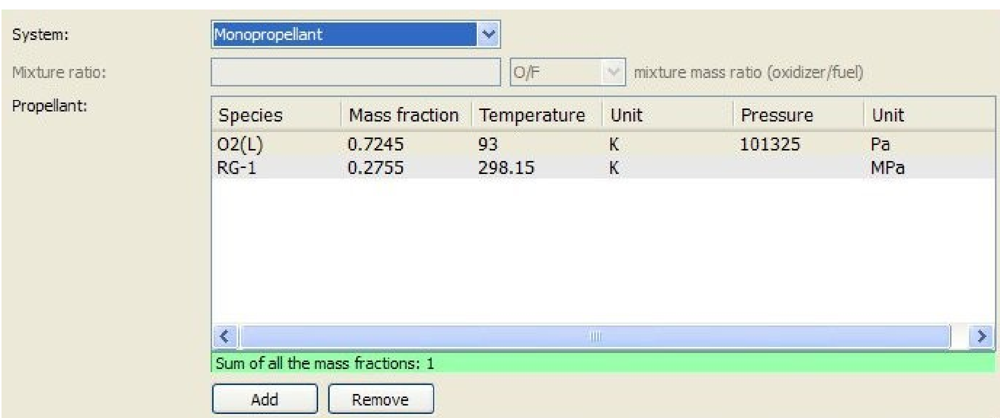
For monopropellant system, the single list of propellant components is enabled, whereas the Fuel list and fields for specifying mixture ratio are disabled.



Specifying monopropellant system

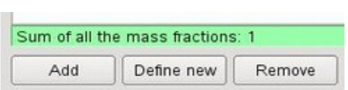
Note: for the thermodynamic calculation, the only difference between bipropellant system and monopropellant system that contains two species, is how the species mixture is defined. Actually, you can define any bipropellant system (as well as three- or more propellant systems) as a monopropellant system, specifying the proper mass fraction for each component on the list.

For instance, the following "monopropellant" configuration is equivalent to the bipropellant one with O/F=0.7245/0.2755=2.63:

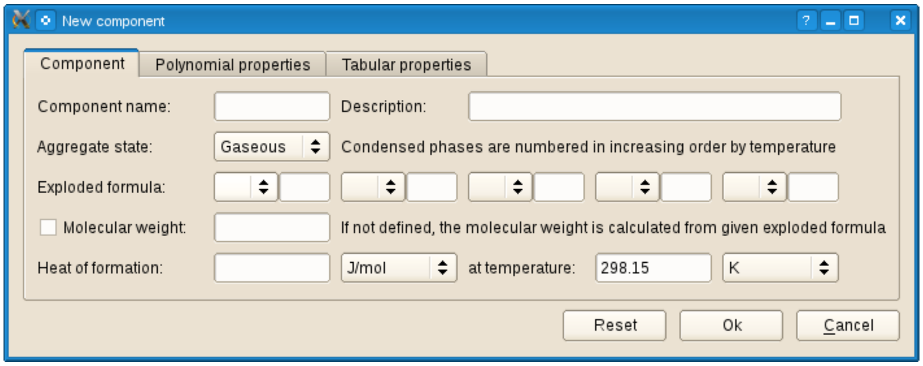


Each component list (Oxidizer, Fuel, or Propellant) contains one or more species, displayed on the single row. To add species to the list, click the button **Add** at the bottom of the corresponding list. To remove the selected species from the list, click the button **Remove**.

To add new species, click the button **Define new** (available in RPA Standard Edition only):



In the appeared dialog window, specify at least the name, the exploded chemical formula, and the heat of formation of new species:



Once the new species was defined, it is stored in the user thermodynamic database and permanent available for using in other problems.

Component Properties

The component on the list features 4 parameters: species name, mass fraction of the species in the component (for bipropellant systems) or propellant (for monopropellant systems), initial temperature of the species, and initial pressure of the species:

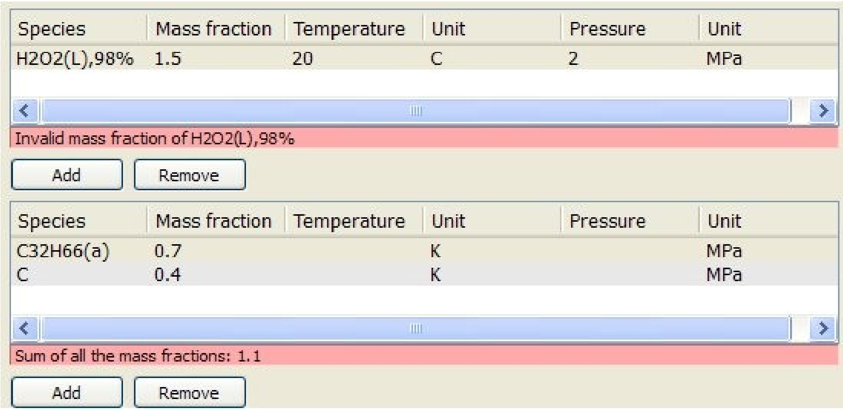
Species	Mass fraction	Temperature	Unit	Pressure	Unit
H2O2(L),98%	1	20	C	2	MPa

Initial species temperature and pressure are optional parameters. If not specified, the following default values will be assigned automatically:

- `p = 1 atm`, `T = 298.15 K` for non-cryogenic species
- `T` = [boiling point temperature] for cryogenic liquids

When composing component (for bipropellant systems) or propellant (for monopropellant systems) from several species, the sum of all the mass fractions of components on the same list has to be equal to 1. To change the mass fraction for the species, double-click on the corresponding cell, enter the new value and press Enter button (or click away).

Each list features the automatic mass fraction checker, that displays the current sum in the list footer. If the sum is correct, the background color of the footer is light-green, otherwise the color is light-red:



To specify the initial temperature and/or pressure of the species, double-click on the corresponding cell, enter the new value and then press Enter button (or click away):

To change the unit, double-click on the corresponding cell, select the desired unit on the list, and then press Enter button (or click away):

The temperature can be entered using one of the following units: `K`, `C`, `F`, `R`.

The pressure is an absolute pressure and can be entered using one of the following units: `MPa`, `atm`, $\texttt{kg/sm}^2$, `bar`, `psi`, `Pa`.

Note: the initial temperature and/or pressure can only be specified for the components which are supplied together with thermodynamic properties either in the polynomial form or in tabular form.

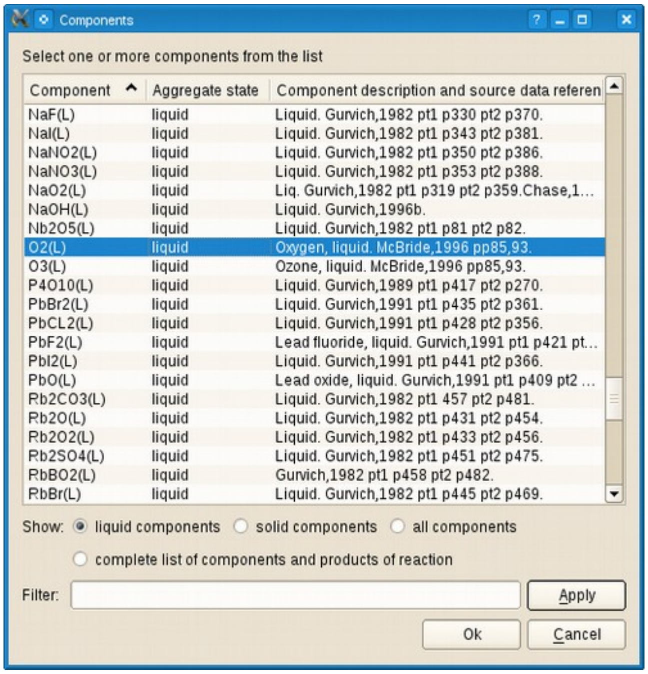
See chapter Thermodynamic Database Editor for further details.

Note: if the components change their temperature/pressure due to the work performed by components themselves, you should assign initial `(T,p)` which components had *before the work is performed*.

For instance, in staged combustion engine, a turbopump is powered by components, and the correct initial parameters correspond to components' conditions at the pump *inlets*, in opposite to the conditions at pump outlets or combustion chamber injector.

Components Database

After clicking on button **Add**, the dialog window *"Components"* appears. The content of the dialog window depends on the type of target list, where component will be added.



Components Database

When adding new component to the oxidizer list, the dialog window displays available oxidizers; when adding new component to the fuel list, the dialog window displays available fuels. For monopropellant systems, both oxidizers and fuels will be displayed in the dialog window.

You can filter the list in the dialog window, using a regular expression. The filter pattern is applied to both columns of the table.

Mark the check box *"Show complete list of available reactants and product of reaction"* if you want to see all species, including atomized and/or ionized products of reaction, or keep it unmarked if you want to see only possible propellant components.

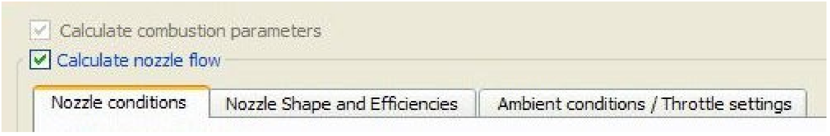
Select one or more species on the list and click the button **OK**. Click the button **Cancel** if you want to leave without adding any species.

Nozzle Flow Model

Nozzle Flow Model Screen

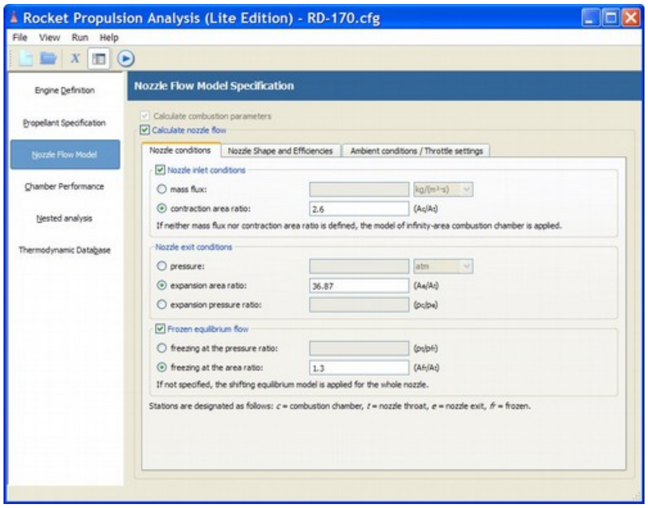
The program can either calculate the combustion parameters in the combustion chamber, or perform the complete engine performance analysis, calculating the flow through the nozzle.

Clear the check box *"Calculate nozzle flow"*, in order to choose the first possibility, or mark it to start the nozzle flow analysis:

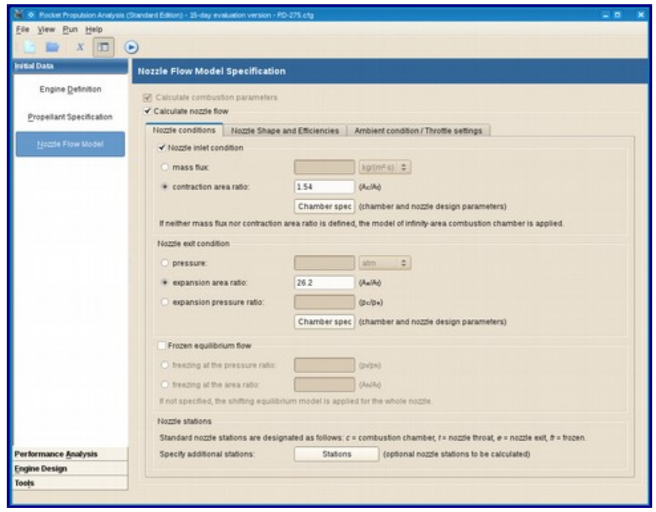


Nozzle conditions

If you are solving the nozzle flow problem, you have to define at least nozzle exit conditions, specifying one of three parameters: nozzle exit pressure, nozzle expansion area ratio, or nozzle expansion pressure ratio.



Nozzle conditions (Lite Edition)



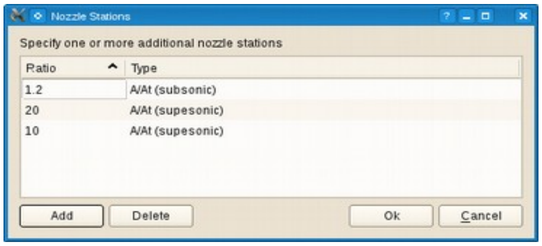
Nozzle conditions (Standard Edition)

The program can calculate the performance for a combustion chambers with finite or infinite cross section area. The default model assumes the infinite cross section area of the chamber. To switch to the finite-area model, mark the check box *"Nozzle inlet conditions"*, and then specify the chamber contraction area ratio `Ac/At` or mass flux in the combustion chamber.

If specified, the nozzle inlet and exit conditions are used for chamber and nozzle sizing, together with additional parameters on the screen Chamber Geometry (press the button **Chamber spec** to jump directly to the that screen).

The program can calculate the performance with respect to the shifting and frozen chemical equilibrium in the nozzle. The default model assumes the shifting chemical equilibrium in the whole nozzle. To switch to the frozen chemical equilibrium model, mark the check box *"Frozen equilibrium flow"*, and then specify the nozzle section, where application of this model should be started.

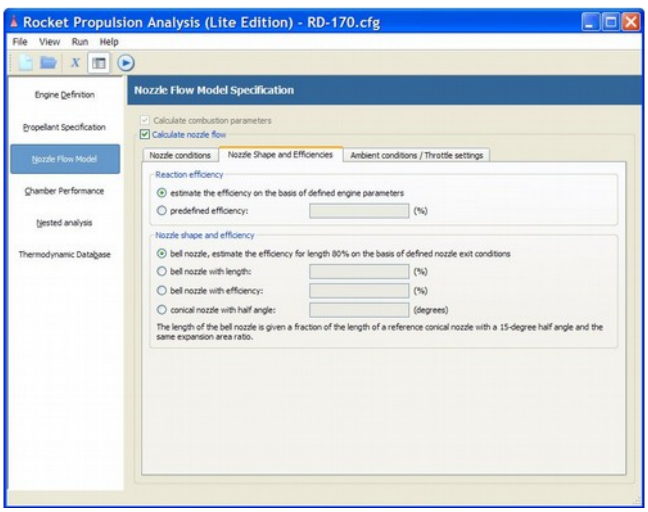
By default, the program calculates parameters at following nozzle stations: nozzle inlet, nozzle throat and nozzle exit. To specify additional nozzle stations, press the button **Stations** and specify one or more additional stations, defining them by area ratio `Ac/At` or by pressure ratio `pc/p`:



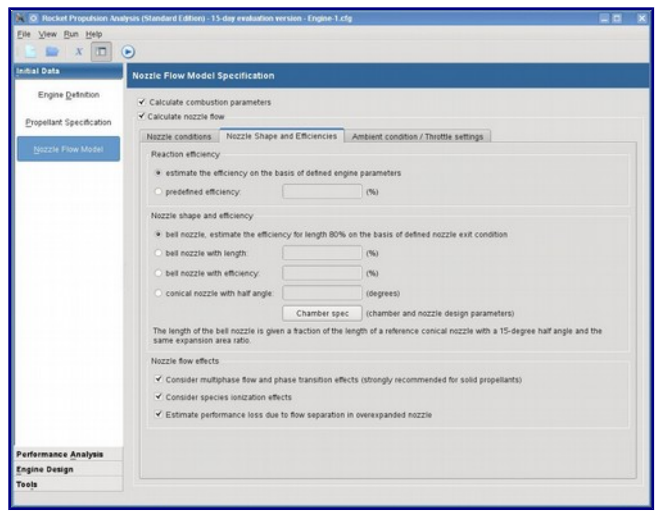
Nozzle Stations (Standard Edition)

Nozzle Shape and Efficiencies

The program calculates both theoretical and delivered engine performance. You can define correction factors on the screen *Nozzle Shape and Efficiencies*, otherwise the program estimates it on the basis of defined engine parameters, such as chamber pressure, propellant components and nozzle conditions.

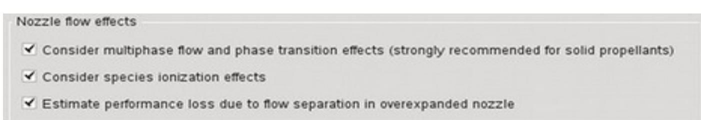


Nozzle Shape and Efficiencies (Lite Edition)



Nozzle Shape and Efficiencies (Standard Edition)

In RPA Standard Edition, you can also control the considered by the program nozzle flow effects:



Switch off the flag **Consider multiphase flow and phase transition** in order to suppress the calculation of multiphase flow effects. Note that for the most of solid propellant problems this flag should be switched on.

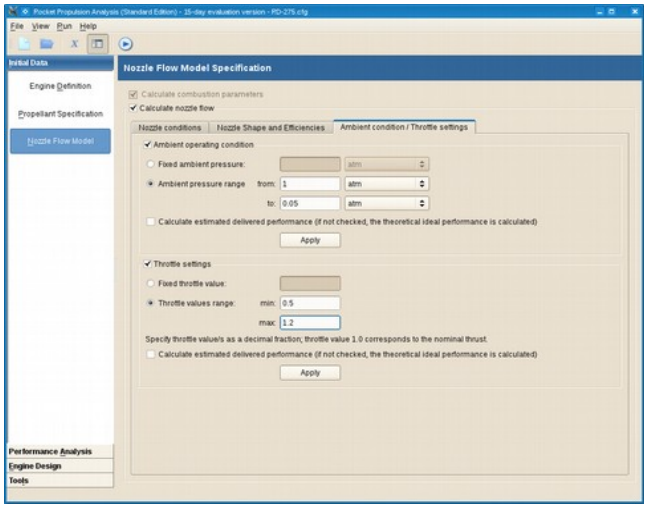
Switch off the flag **Consider species ionization effects** in order to to suppress the calculation of species ionization effects.

Switch off the flag **Estimate performance loss due to flow separation** in order to disable the additional calculation of flow separation effects.

Ambient condition

By default the program calculates performance of the rocket engine at the sea level conditions (`pa=1 atm`, or `14.7 psi`), optimum nozzle expansion (`pe=pa`), and vacuum conditions (`pa=0`).

To calculate the performance at desired ambient conditions, you can also explicitly specify either the specific ambient pressure or the range of ambient pressures given as high and low range values.



Ambient condition

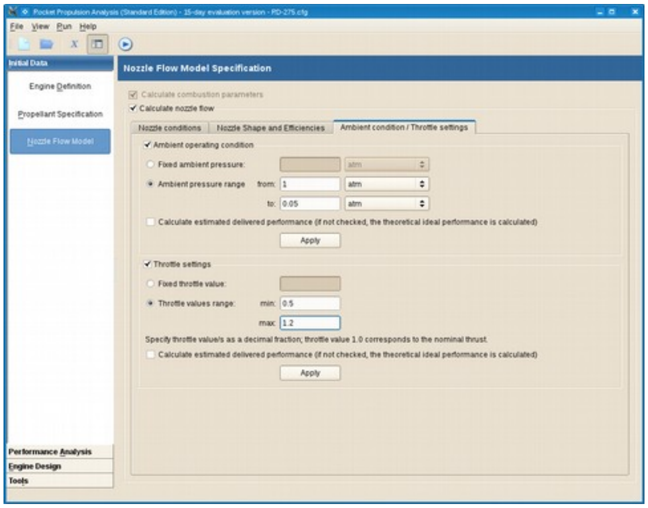
Switch on the flag “Calculate estimated delivered performance” in order to apply the performance corrections factors to results of calculation.

The pressure is an absolute pressure and can be entered using one of the following units: `MPa`, `atm`, `kg/sm`^{2}, `bar`, `psi`, `Pa`.

Throttle settings

By default the program calculates performance of the rocket engine assuming the propellant flow rate that correspond to nominal thrust.

To calculate the performance at desired throttle settings, you can also explicitly specify either the specific throttle value, or the range of throttle values given as high and low range values.



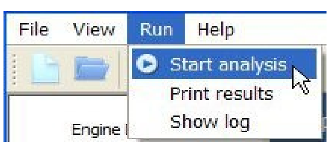
Throttle settings

Switch on the flag "Calculate estimated delivered performance" in order to apply the performance corrections factors to results of calculation.

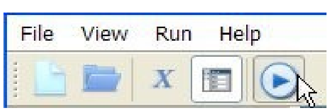
The throttle value is a ratio of propellant flow rate at desired throttle setting to flow rate that correspond to nominal thrust (100% flow rate).

Starting Analysis

After specifying the initial data, or opening existing configuration file, you can start the analysis by clicking menu **Run**, and then **Start analysis** in menu bar:

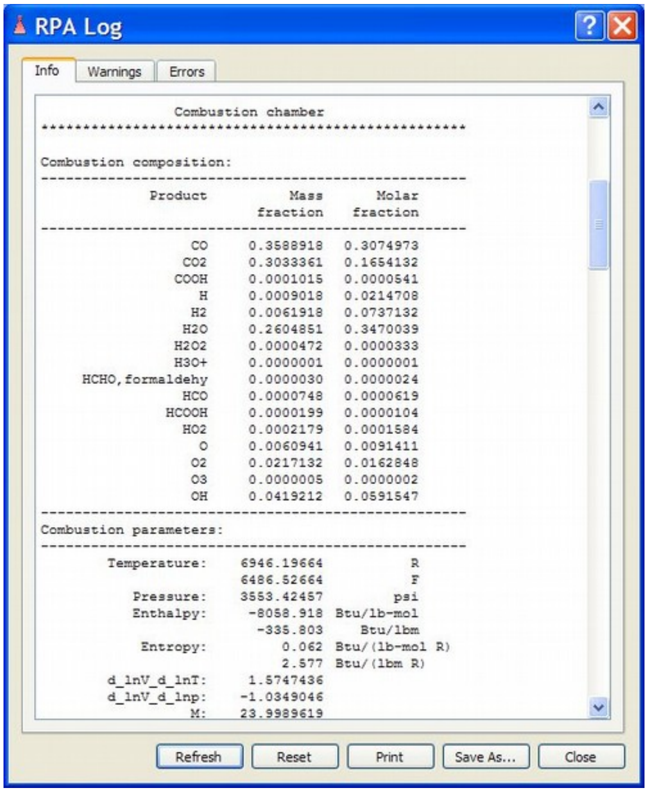


or clicking icon **Start** on the main toolbar:



After successful finishing the analysis the program automatically switches to the screen Chamber Performance, that displays the calculated results.

You can also print out the results, clicking menu **Run**, and then **Print results**, or check the analysis log, clicking menu **Run**, and then **Show log** in menu bar. The analysis log as an ASCII file is also available in the user directory.



Analysis Log

Chamber Performance

The screen Chamber Performance consists of 4 tabs *Thermodynamic properties*, *Performance*, *Altitude performance*, and *Throttled performance*.

You can print out the results, clicking the button **Print**, or save the results as ASCII or HTML file, clicking the button **Save As...** at the right-bottom corner of the screen.

Propellant properties

The following propellant properties are available in the analysis log:

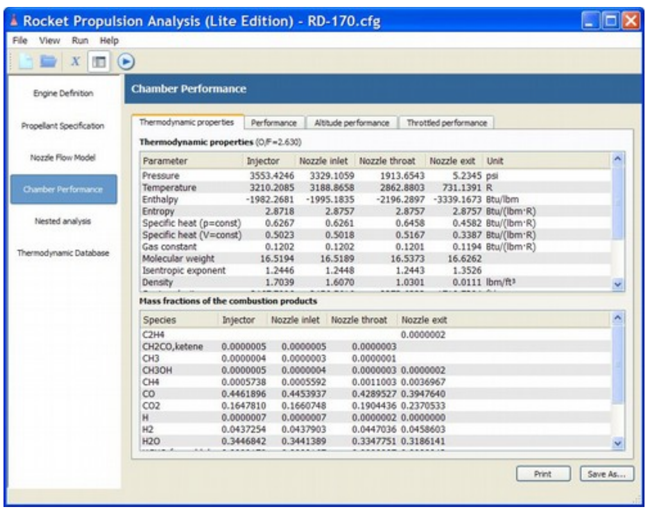
- propellant exploded formula,
- oxidizer excess coefficient,
- stoichiometric mixture ratio,
- propellant density (only if density values of all components are available).

```
Propellant:
---------------------------------------------------
    Component  Temp.    Pressure        Mass
                [K]       [MPa]       fraction
---------------------------------------------------
        H2(L)   20.3     0.1013     0.1428571
        O2(L)   90.2     0.1013     0.8571429
---------------------------------------------------
Propellant exploded formula: (O)0.549 (H)1.451
              alpha:     0.7559833 (oxidizer excess coefficient)
                O/F:     6.0000000
              O/F 0:     7.9366827 (stoichiometric)
                rho:     932.86285      kg/m^3
```

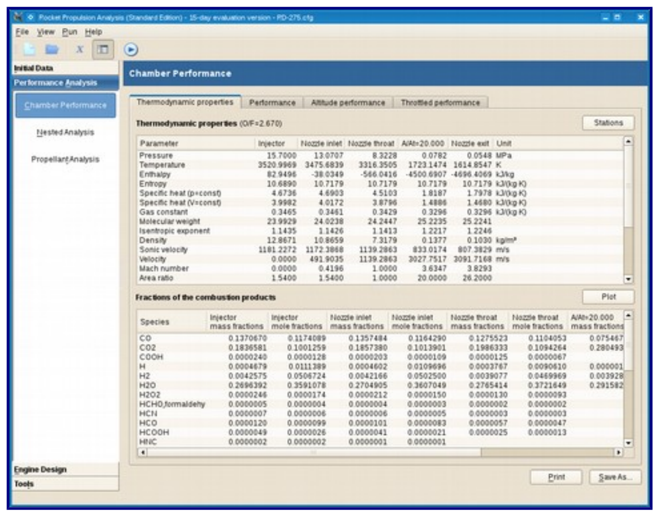
Propellant properties in the analysis log

Thermodynamic properties

The tab *Thermodynamic properties* displays the parameters of reaction products and their mass fractions at the chamber stations involved into the analysis.



Thermodynamic properties (Lite Edition)



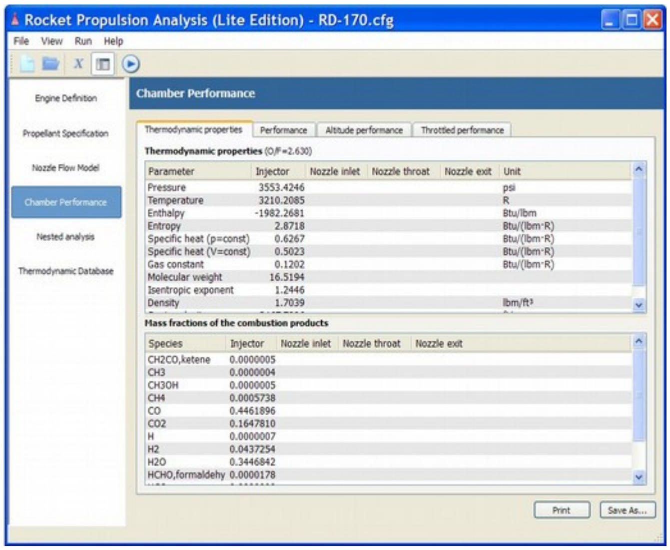
Thermodynamic properties (Standard Edition)

The current propellant components ratio (either explicitly defined on the screen Propellant Specification, or found by optimizer) is displayed in the header of the top table on the tab:

If the problem is configured to solve the parameters in the combustion chamber (see chapter Nozzle Flow Model), the only available nozzle station is *Injector* (see figure *Injector thermodynamic properties*), otherwise the available chamber stations are *Injector*, *Nozzle inlet*, *Nozzle throat* and *Nozzle exit* as well as additional stations, defined on screen *Nozzle Flow Model* (see figure *Thermodynamic properties above*).

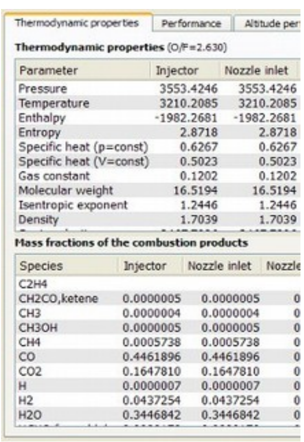
You can specify additional nozzle stations either on screen Nozzle Flow Model, or on screen

Thermodynamic properties, pressing the button **Stations**.



Injector thermodynamic properties

If the problem is configured to calculate the performance for the combustion chamber with infinite cross section area, the parameters for *Nozzle inlet* station are identical to that at *Injector* station:

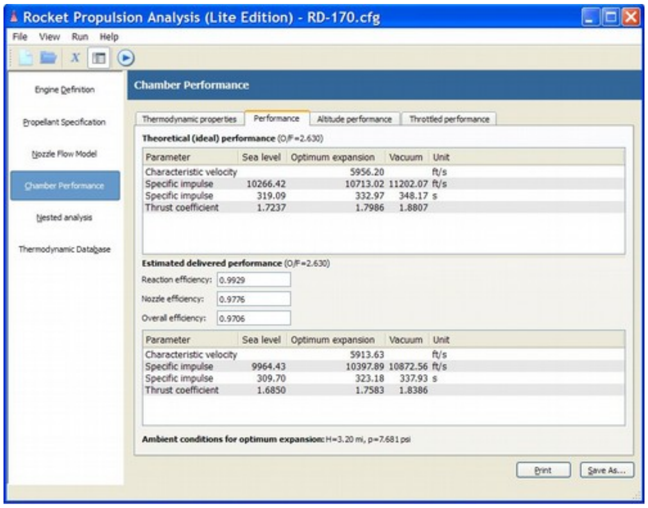


The thermodynamic parameters are displayed in the current units, defined in the Preferences Dialog.

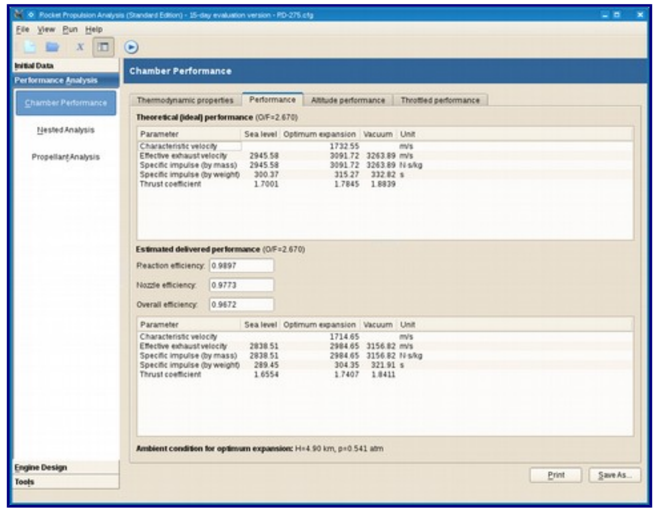
You can shrink or heighten the top table while the bottom table will be heightened or shrunk, dragging the horizontal bar between the tables down or up.

Performance

The tab *Performance* displays the theoretical (ideal) performance of the chamber, as well as its estimated delivered performance and the correction factors used to calculate the delivered performance.



Chamber performance (Lite Edition)



Chamber performance (Standard Edition)

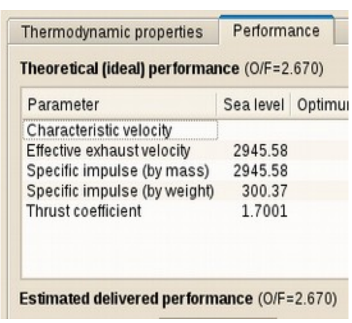
Note: if you are analyzing a pressure-fed-cycle or staged-combustion-cycle engine, the calculated performance is actually an *engine* performance.

If the thrust chamber sizing parameters have been configured (see screen Engine Definition), the program calculates the divergence thrust loss and displays it on the screen Thrust Chamber Size and Geometry:

```
   De =   58.44  mm
e_div = 0.00478  (divergence loss)
```

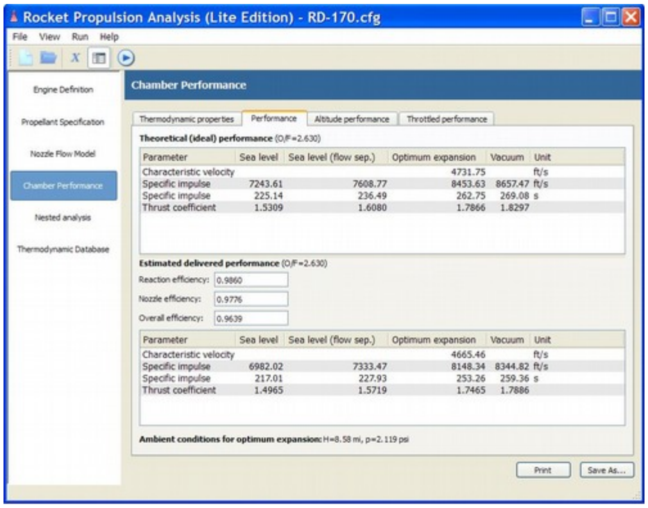
After the first program run, you can use the calculated value of divergence thrust loss as well the calculated value of friction thrust loss (see Thermal Analysis) to adjust the nozzle efficiency on the screen Nozzle Shape and Efficiencies.

The current propellant components ratio (either explicitly defined on the screen Propellant Specification, or found by optimizer) is displayed in the header of the both tables on the tab:



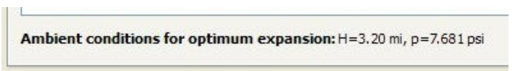
If the problem is configured to solve the combustion parameters at *Injector* station (see chapter Nozzle Flow Model), the performance parameters are not available, otherwise the tab displays calculated performance of the chamber at the sea level conditions (`pa=1 atm`, or `14.7 psi`), optimum nozzle expansion (`pe=pa`), and vacuum conditions (`pa=0`).

In case if flow separation occurs in the nozzle, the additional column displays the performance at sea level with respect to the flow separation.



Chamber performance with flow separation

Ambient conditions for optimum nozzle expansion (altitude and ambient pressure) are displayed at the bottom of the tab:

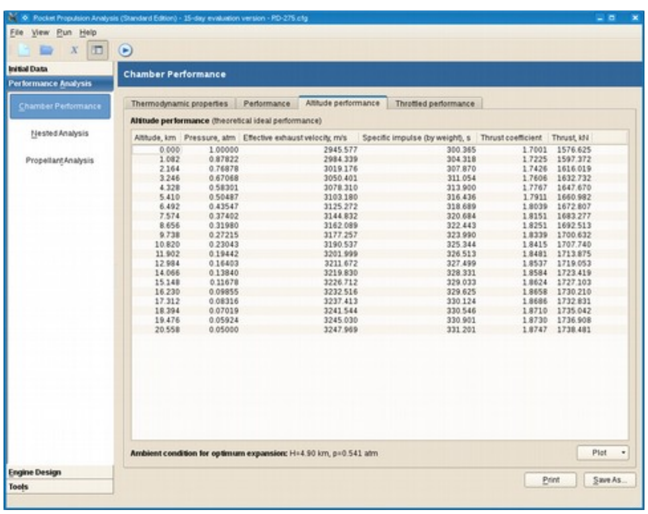


The performance parameters are displayed in the current units, defined in the Preferences Dialog.

You can shrink or heighten the top table while the bottom table will be heightened or shrunk, dragging the horizontal bar between the tables down or up.

Altitude performance

The tab *Altitude performance* displays the performance of the chamber in the specified ambient conditions.



Altitude performance

By default, the program calculates ideal theoretical performance. If you want to calculate estimated delivered performance, switch on the flag "Calculate estimated delivered performance" on screen *Ambient Conditions*.

If the problem is configured to solve the combustion parameters at *Injector* station (see chapter Nozzle Flow Model), the performance parameters are not available, otherwise the tab displays calculated performance of the chamber at the defined altitude and ambient pressure.

You can plot the diagrams *"specific impulse vs. altitude"*, *"thrust coefficients vs. altitude"* or *"thrust vs. altitude"* clicking the button **Plot** at the bottom-right corner of the tab. The altitude of the optimum expansion, as well as the altitude of flow separation (if occurs) is shown on the diagram by corresponded vertical marker line/s.

Plot - altitude performance

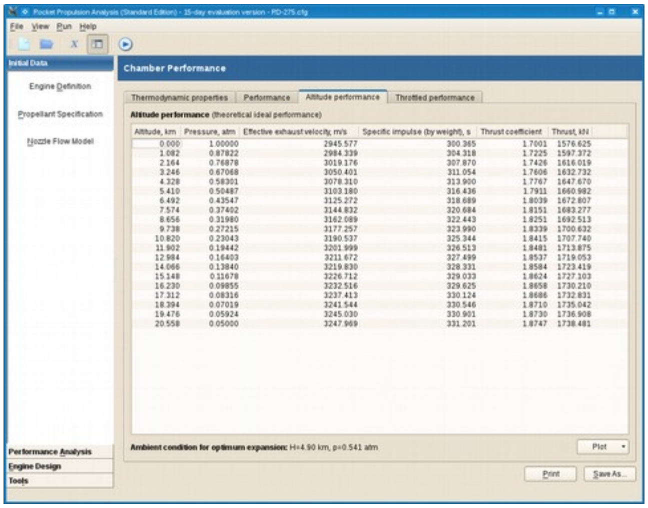
Plot - altitude performance with flow separation

Ambient conditions for optimum nozzle expansion (altitude and ambient pressure) are displayed at the bottom of the tab.

The performance parameters are displayed in the current units, defined in the Preferences Dialog.

Throttled performance

The tab *Throttled performance* displays the performance of the chamber at the specified throttle values.



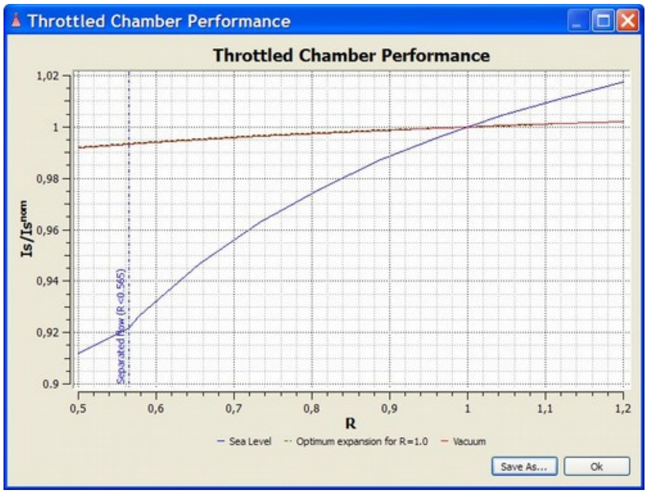
Altitude, km	Pressure, atm	Effective exhaust velocity, m/s	Specific impulse (by weight), s	Thrust coefficient	Thrust, kN
0.000	1.00000	2945.577	300.365	1.7001	1576.625
1.082	0.87822	2984.339	304.318	1.7225	1597.372
2.164	0.76878	3019.176	307.870	1.7426	1616.019
3.246	0.67068	3050.401	311.054	1.7606	1632.732
4.328	0.58301	3078.310	313.900	1.7767	1647.670
5.410	0.50487	3103.180	316.436	1.7911	1660.982
6.492	0.43547	3125.272	318.689	1.8039	1672.807
7.574	0.37402	3144.832	320.684	1.8151	1683.277
8.656	0.31980	3162.089	322.443	1.8251	1692.513
9.738	0.27215	3177.257	323.990	1.8339	1700.632
10.820	0.23043	3190.537	325.344	1.8415	1707.740
11.902	0.19442	3201.999	326.513	1.8481	1713.875
12.984	0.16403	3211.672	327.499	1.8537	1719.053
14.066	0.13840	3219.830	328.331	1.8584	1723.419
15.148	0.11678	3226.712	329.033	1.8624	1727.103
16.230	0.09855	3232.516	329.625	1.8658	1730.210
17.312	0.08316	3237.413	330.124	1.8686	1732.831
18.394	0.07019	3241.544	330.546	1.8710	1735.042
19.476	0.05924	3245.030	330.901	1.8730	1736.908
20.558	0.05000	3247.969	331.201	1.8747	1738.481

Throttled performance

By default, the program calculates ideal theoretical performance. If you want to calculate estimated delivered performance, switch on the flag “Calculate estimated delivered performance” on screen *Throttle Settings* .

If the problem is configured to solve the combustion parameters at *Injector* station (see chapter Nozzle Flow Model), the performance parameters are not available, otherwise the tab displays calculated performance of the rocket engine at the sea level conditions (`pa=1 atm`, or `14.7 psi`), optimum nozzle expansion (`pe=pa`), and vacuum conditions (`pa=0`) at the defined throttle values.

You can plot the diagrams *"specific impulse vs. throttle value"* or *"thrust vs. throttle value"* clicking the button **Plot** at the bottom-right corner of the tab. The throttle value where flow separation occurs is shown on the diagram by corresponded vertical marker line.



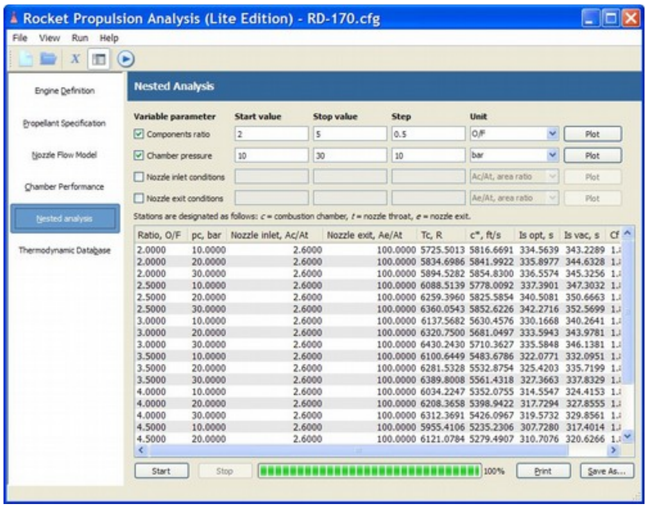
Plot - throttled performance

The performance parameters are displayed in the current units, defined in the Preferences

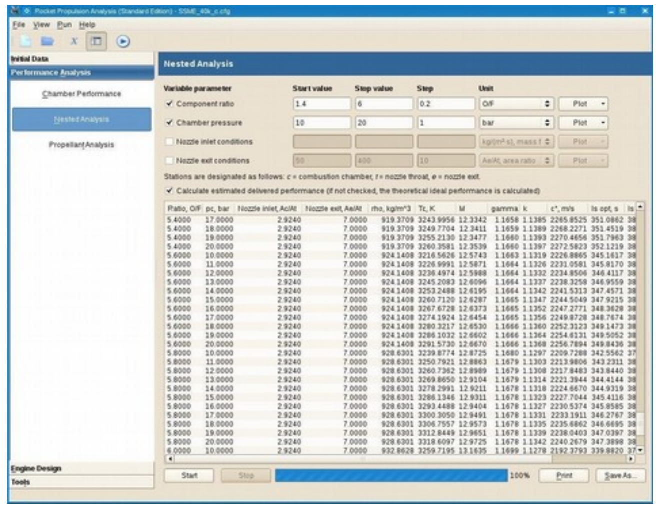
Dialog.

Nested Analysis

Using nested analysis, you can evaluate the performance of the rocket chamber for the range of parameter/s, stepping of up to four independent variables (component ratio, chamber pressure, nozzle inlet conditions, nozzle exit conditions).



Nested analysis (Lite Edition)



Nested analysis (Standard Edition)

Nested analysis inherits the initial configuration parameters, defined on the screens Engine Definition, Propellant Specification, and Nozzle Flow Model. You can define up to four variables parameters, which replace corresponding parameter/s of initial configuration, and start the nested analysis, clicking the button **Start** at the left-bottom corner of the tab.

Note: since the program performs the thermodynamic analysis for each unique combination of variable parameters, definition of parameters with a small step can lead to very long calculation time.

By default, the program calculates ideal theoretical performance. If you want to calculate estimated delivered performance, switch on the corresponding flag:

The program performs the thermodynamic analysis for each unique combination of variable parameters, stepping it by defined value, and displaying the results in the table at the bottom of the tab. The current status of nested analysis is shown by progress bar at the bottom of the tab. The running analysis can be stopped by clicking the button **Stop**.

After finishing the nested analysis, you can plot the diagrams *"specific impulse vs. variable parameter"*, *"chamber temperature vs. variable parameter"*, *"characteristic velocity vs. variable parameter"* or *"thrust coefficient vs. variable parameter"*, clicking the button **Plot** at the right side of the corresponding parameter configuration.

Nested analysis plot

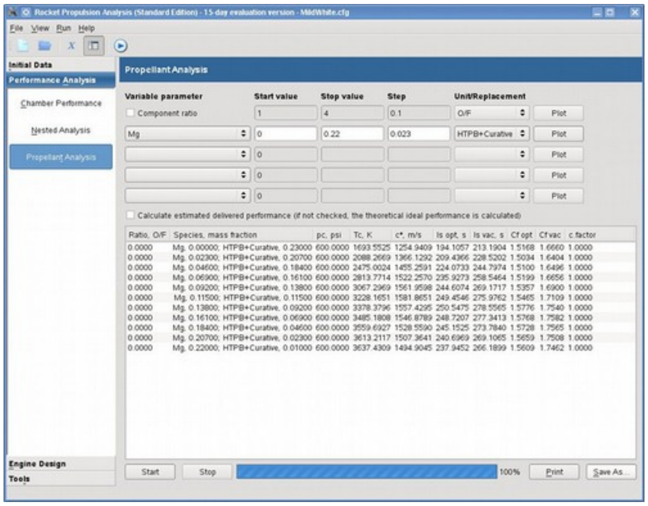
You can print out the results of nested analysis, clicking the button **Print**, or save the results as ASCII or HTML file clicking the button **Save As...** at the right-bottom corner of the screen.

In RPA Standard Edition, you can also save the results in PDF or ODF formats.

The defined parameters of the nested analysis are stored in the global INI-file and shared between different configurations.

Propellant Analysis

Using propellant analysis tool, you can evaluate different propellant compositions, with stepwise replacement of one component with another one.

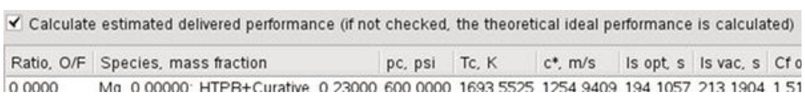


Propellant analysis

Propellant analysis inherits the initial configuration parameters, defined on the screens Engine Definition, Propellant Specification, and Nozzle Flow Model. You can define up to four pairs of component, which replace corresponding components of initial propellant composition, and start the propellant analysis, clicking the button **Start** at the left-bottom corner of the tab.

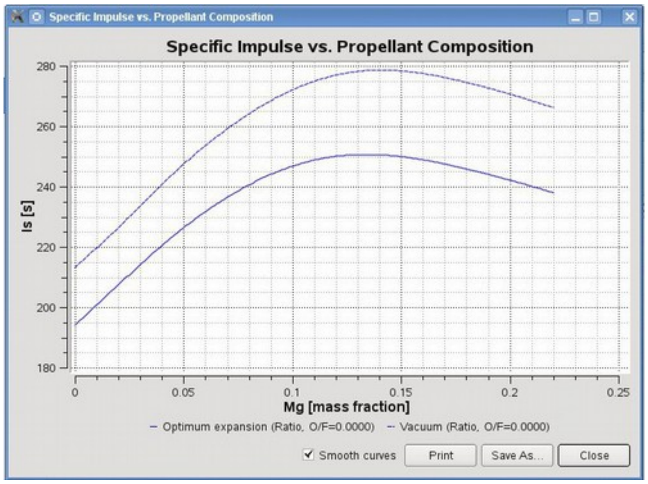
Note: since the program performs the thermodynamic analysis for each unique combination of variable parameters, definition of parameters with a small step can lead to very long calculation time.

By default, the program calculates ideal theoretical performance. If you want to calculate estimated delivered performance, switch on the corresponding flag:



The program performs the thermodynamic analysis for each unique combination of variable parameters, stepping it by defined value, and displaying the results in the table at the bottom of the tab. The current status of propellant analysis is shown by progress bar at the bottom of the tab. The running analysis can be stopped by clicking the button **Stop**.

After finishing the propellant analysis, you can plot the diagrams *"specific impulse vs. variable parameter", "chamber temperature vs. variable parameter", "characteristic velocity vs. variable parameter" or "thrust coefficient vs. variable parameter"*, clicking the button **Plot** at the right side of the corresponding parameter configuration.



Propellant analysis plot

You can print out the results of propellant analysis, clicking the button **Print**, or save the results as ASCII or HTML file clicking the button **Save As...** at the right-bottom corner of the screen.

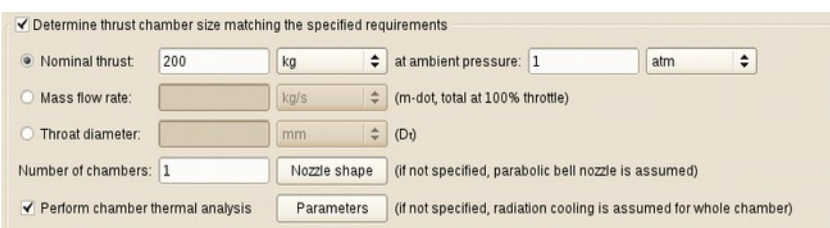
In RPA Standard Edition, you can also save the results in PDF or ODF formats.

The defined parameters of the propellant analysis are stored in the global INI-file and shared between different configurations.

Chamber Geometry

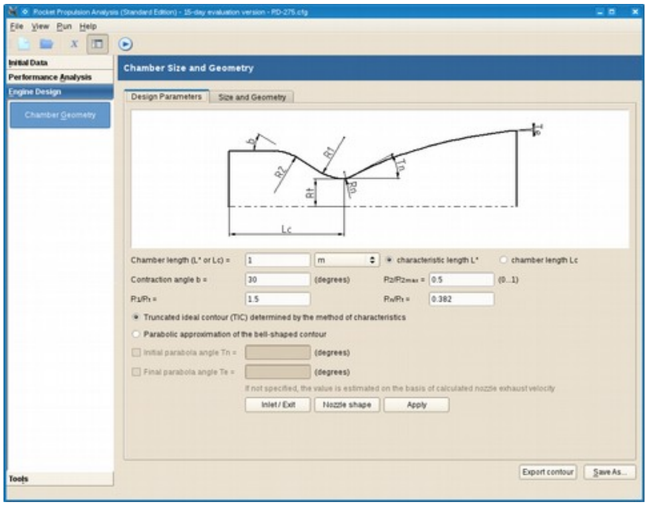
The screen Chamber Geometry consists of 2 tabs *Design Parameters* and *Size and Geometry*.

The input of parameters on the screen is only enabled if the flag "Determine thrust chamber size" on the screen Engine Definition is switched on, and at least one of sizing parameters is provided:



Design Parameters

On the tab *Design Parameters* you can define design parameters used to calculate the size of the combustion chamber and the shaped of the nozzle.

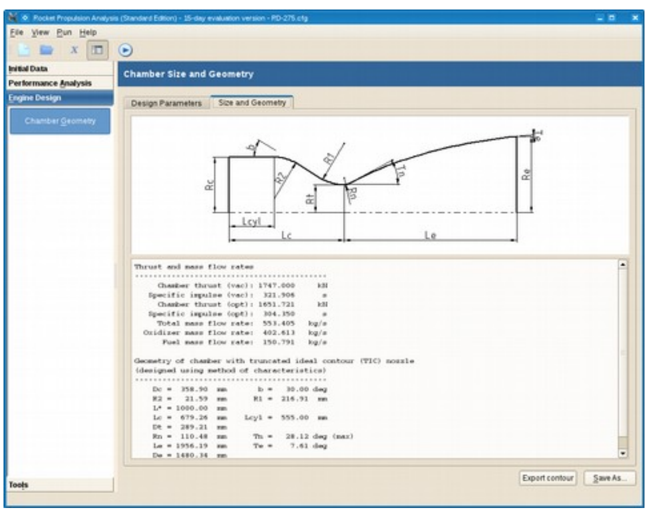


Design parameters

Note: such design parameters as nozzle inlet and exit conditions, as well as a nozzle shape (bell or conical) can be only changed on screen Nozzle Flow Model. Use provided buttons to jump direct to that screen and back.

Size and Geometry

The tab *Size and Geometry* displays the calculated size of the chamber and nozzle.



Size and geometry

If the thrust chamber sizing parameters have been configured (see screen Engine Definition), the program calculates the divergence thrust loss and displays it on the screen Thrust Chamber Size and Geometry:

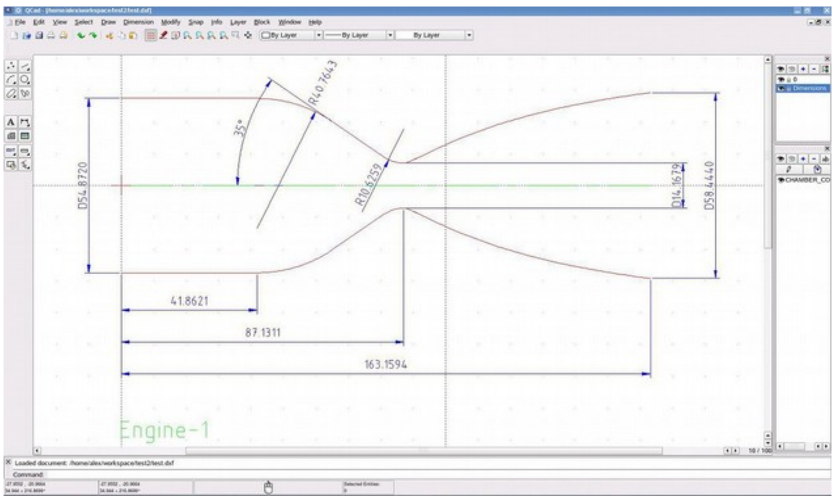
```
    De =    58.44  mm
e_div = 0.00478   (divergence loss)
```

After the first program run, you can use the calculated value of divergence thrust loss as well the calculated value of friction thrust loss (see Thermal Analysis) to adjust the nozzle efficiency on the screen Nozzle Shape and Efficiencies.

You can save the results as an ASCII file, clicking the button **Save As...**, or export the contour to the DXF file clicking the button **Export to DXF** at the right-bottom corner of the screen.

If exporting to DXF file, you provide additional export parameters:

The resulting DXF file can be opened in any CAD program.



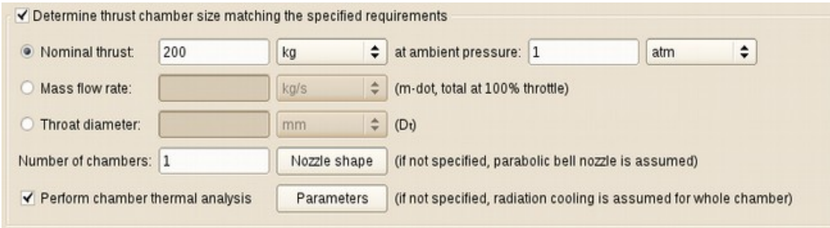
DXF file opened in QCad

Thermal Analysis

The program is capable of performing the steady-state thermal analysis.

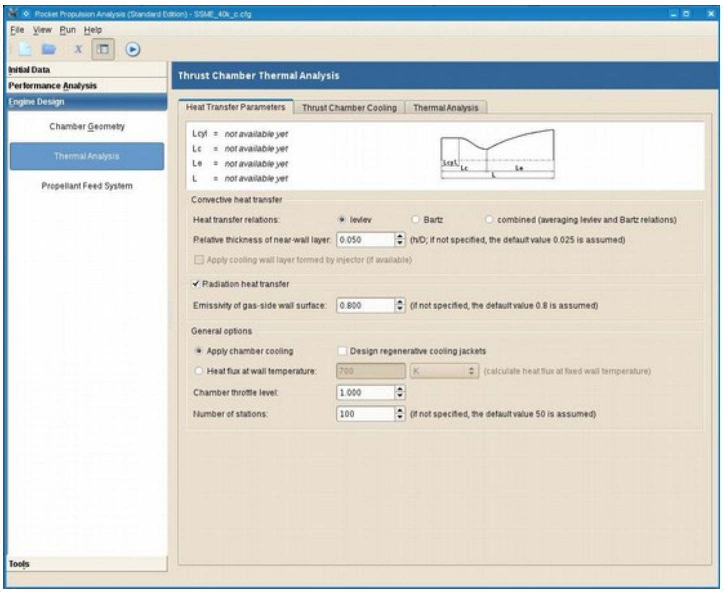
The screen Thermal Analysis consists of 3 tabs *Heat Transfer Parameters*, *Thrust Chamber Cooling* and *Thermal Analysis*.

The input of parameters on the screens are only enabled if both flags "Determine thrust chamber size" and "Perform chamber thermal analysis" on the screen Engine Definition are switched on, and at least one of sizing parameters is provided:



Heat Transfer Parameters

On the tab *Heat Transfer Parameters* you can define heat transfer parameters used to calculate the heat transfer rate distribution in the thrust chamber.



Heat transfer parameters

The program implements two different methods of calculating the gas-side heat transfer rates: Ievlev approach for calculation of convective heat transfer in the nozzle and Bartz semi-empirical correlation for gas-side heat transfer coefficient.

The user has the possibility to choose either one of two mentioned methods or combined approach, which calculates the resulting heat transfer rate as an average value of that calculated by Ievlev and Bartz methods.

The program can perform the thermal analysis, calculating the heat flux either as a convective heat transfer only or summing the contributions of convective and radiation heat transfers. In the last case, the user shall specify a radiation emissivity coefficient of the wall inner surface.

Option *Heat flux at wall temperature* allows to estimate the heat flux distribution at specified constant temperature of the gas-side wall surface.

Option *Apply chamber cooling* allows to estimate the temperature and heat flux distribution, applying the specified cooling (see chapter *Thrust Chamber Cooling*). When option *Design regenerative cooling jackets* is activated, the parameters of cooling jackets (such as diameter and number of tubes, or size and number of channels) will be automatically adjusted. Note that you still need to define the location, type and connection of cooling jackets, thermal conductivity of the wall, and thickness of inner wall (or tube wall).

After running the thermal analysis, the obtained parameters of cooling jackets are available for editing under the tab *Thrust Chamber Cooling*.

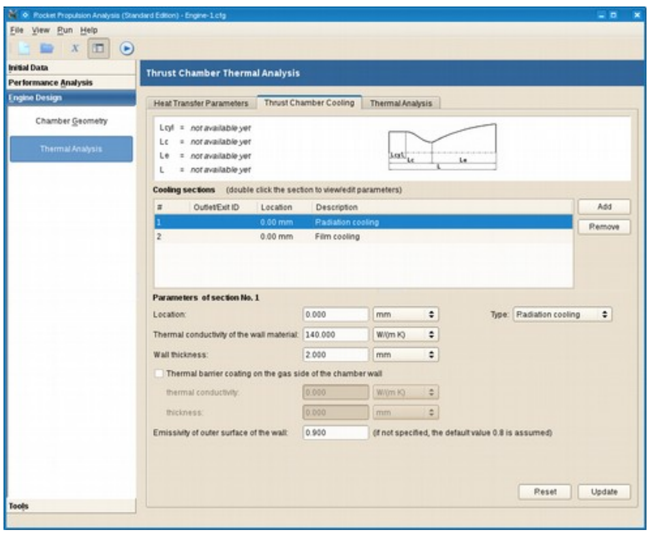
Option *Design regenerative cooling jackets* is not stored in configuration file and will be automatically deactivated after finishing the thermal analysis.

The parameter *Chamber throttle level* defines the throttle level of the engine. This parameter affects the conditions in combustion chamber and mass flow rate through the cooling jackets.

The parameter *Number of stations* defines the number of stations evenly distributed along the thrust chamber. In addition, the program automatically inserts stations at nozzle inlet and nozzle throat, as well as at location of film cooling slots and cooling sections (see *Thrust Chamber Cooling*).

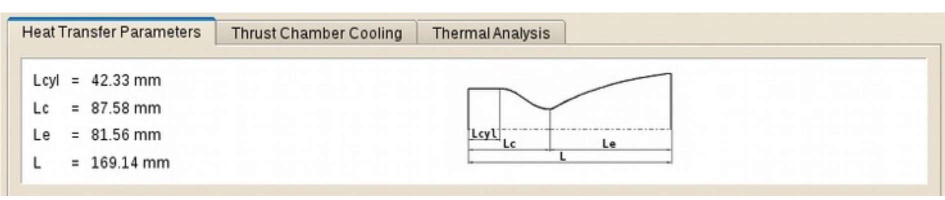
Thrust Chamber Cooling

On the tab *Thrust Chamber Cooling* you can define design parameters of thrust chamber cooling.



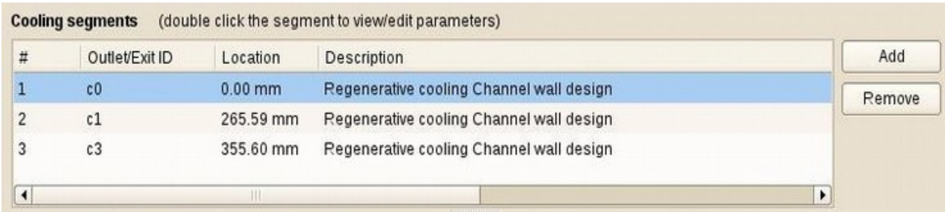
Thrust chamber cooling

In order to make easier the definition of location of cooling sections or film slots, the program displays the dimensions of thrust chamber:



Note that the dimensions are only available after the first run of the program.

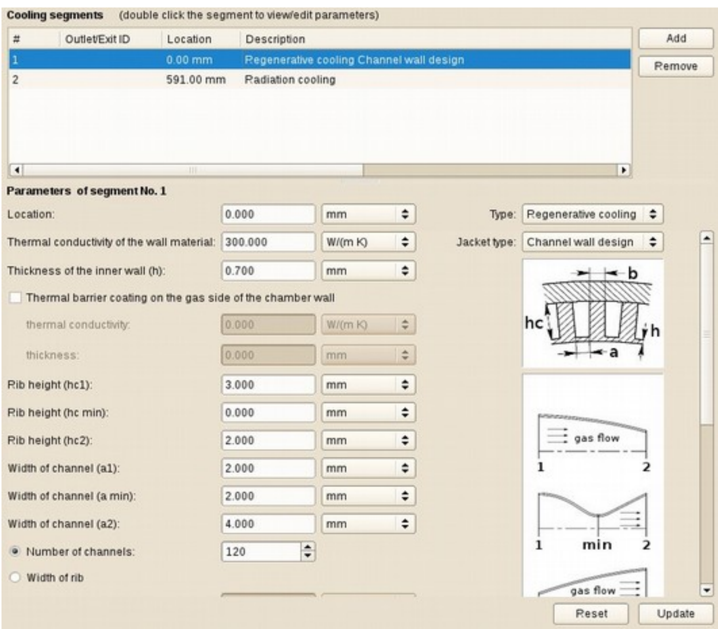
The table *Cooling segments* displays all cooling segments configured for specific thrust chamber, as well as its ID (if available), location and short description:



The user can add new segment pressing the button **Add** or delete existing segment pressing the button **Remove**.

Note: all new cooling segments are created as "Radiation cooling". If you need another type of cooling, double click the created segment and change the type in the parameters area.

To view all parameters assigned to of the existing cooling segment, or to change the type of the segment or modify its parameters, double-click the segment on the list. All parameters of the segment will be displayed in the parameters area just below the table:



The number of currently edited segment is displayed at the bottom of the parameter area.

Use the button **Update** in order to re-assign the modified parameters to edited segment.

Use the button **Reset** in order to reload the parameters from the segment. Note that you will loss all changes in the parameters area.

The user has to define the location of the each cooling segment or film cooling, specifying the distance from the injector plate. The length of the segment is defined implicitly by the location of the next cooling segment or nozzle exit (if no further segment is available).

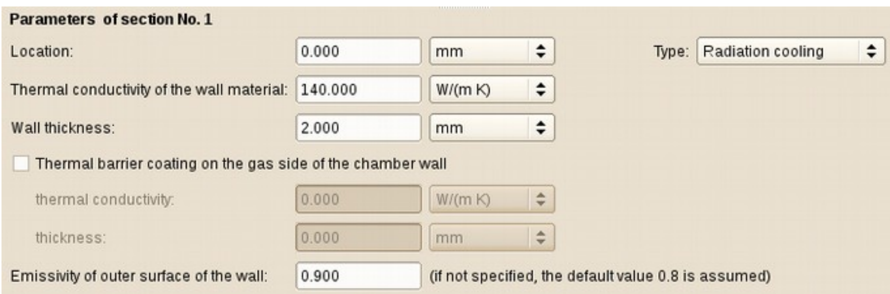
The program supports 3 types of chamber cooling: radiation cooling, regenerative cooling and film cooling.

Radiation Cooling

A radiation cooled chamber transmits the heat from its outer surface, chamber or nozzle extension. Radiation cooling is typically used for small thrust chambers with a high-temperature wall material and/or in low-heat flux regions, such as a nozzle extension.

Configuration of the radiation cooling section includes the following parameters:

• location of the section	required
• wall thickness and thermal conductivity	required
• thermal barrier coating layer thickness and thermal conductivity (see also Thermal Barrier Coating Layer)	optional
• radiation emissivity coefficient of the wall outer surface	required



Radiation cooling parameters

Radiation cooling segment can be combined with the film cooling.

Note: the current version of the program may fail to calculate the thermal state of radiation cooling segment with additional film cooling, if calculation of the radiation heat transfer is disabled (see screen "*Heat Transfer Parameters*").

Regenerative Cooling

Regenerative cooling is the most widely used method of cooling a thrust chamber and is accomplished by flowing high-velocity coolant over the back side of the chamber hot gas wall to convectively cool the hot liner. The coolant with the heat input from cooling the liner is then usually discharged into the injector and utilized as a propellant.

Configuration of the regenerative cooling segment includes the following parameters:

• location of the section	required
• type of design	required
• section ID	required if the exit from the segment connected with input of another regenerative cooling

	segment
• wall thickness and thermal conductivity	required
• thermal barrier coating layer thickness and thermal conductivity (see also *Thermal Barrier Coating Layer*)	optional
• coolant definition: component/s, initial temperature and pressure, relative mass flow rate *or* connection with exit of another regenerative cooling segment, specified by its ID	required
• flag of opposite flow direction	optional
• flag of two-pass coolant flow	optional

Regenerative cooling segment can be combined with the film cooling.

The program supports 3 types of regenerative cooling design: coaxial shell jacket design, tubular wall jacket design and channel wall jacket design.

Coolant definition

The list of coolant components contains one or more species, displayed on the single row. To add species to the list, click the button **Add** at the bottom of the corresponding list. To remove the selected species from the list, click the button **Remove**.

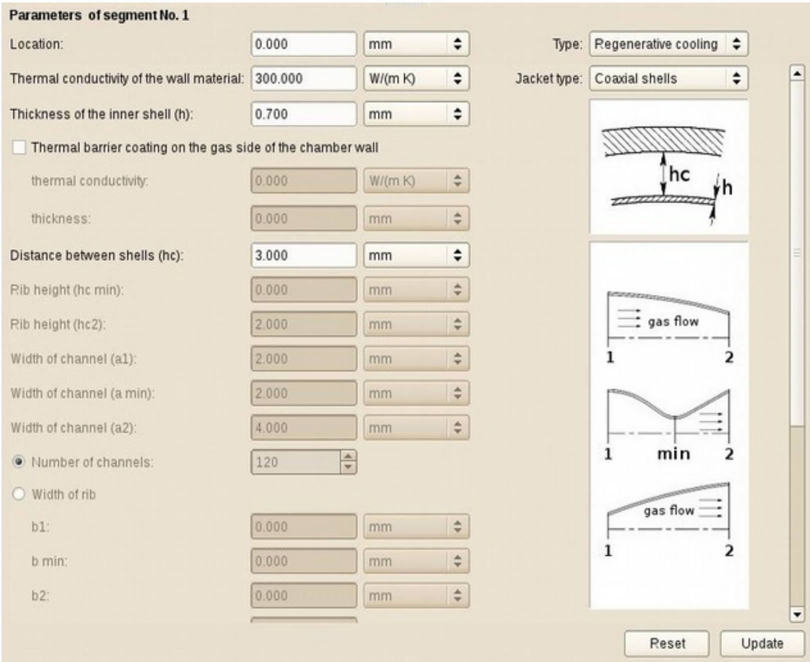
Note: only use the components available in the properties, because the extended NASA database does not include all required properties such as viscosity, thermal conductivity, density, temperature of vaporization/decomposition and heat of vaporization of liquid components.

Alternatively, you can configure the connection between the inlet of the current segment with the exit of another segment, specifying its ID. In this case the coolant component/s and the mass flow rate are exactly the same as defined for the first segment, whereas the initial temperature and pressure are equal to temperature and pressure at exit of the first segment, calculated during the thermal analysis.

Coaxial Shell Jacket Design

Design-specific configuration of the segment includes the following parameters:

• distance between inner and outer walls	required



Coaxial shell jacket parameters

The program supports the segments with constant distance between the walls. If you need to define the chamber with variable inter-wall distance, you have to define several segments each with different distance.

Typical mistakes during the analysis run:

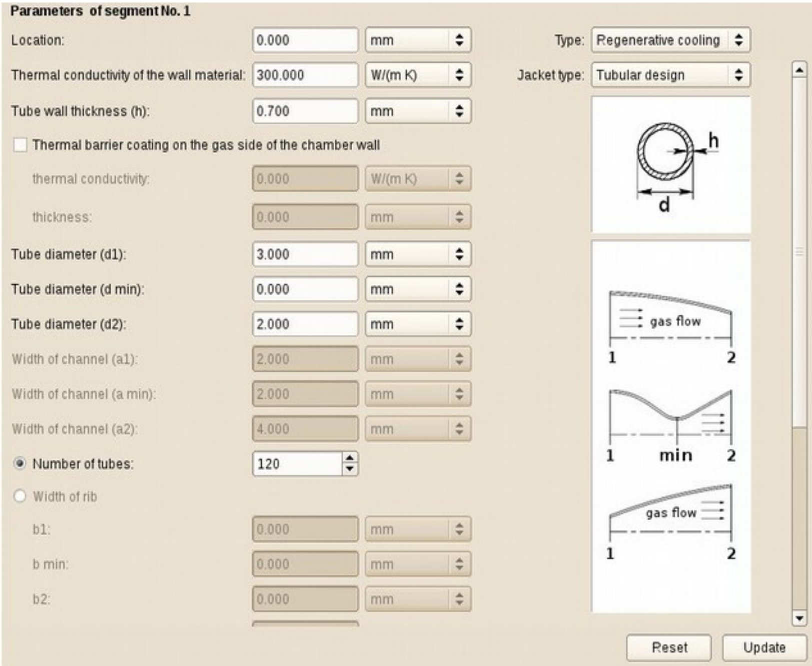
- too small inter-wall distance or too large coolant mass flow rate lead to high pressure drop; the message "Negative pressure drop" appears;
- due to too large inter-wall distance or too small mass flow rate the program cannot converge the solution.

Possible solution: change one or several relevant parameters.

Tubular Wall Jacket Design

Design-specific configuration of the segment includes the following parameters:

• diameter of tube "*d1*", "*d2*" at both sides of the segment	required
• diameter of tube "*d min*" at minimal diameter of the segment	optional; can be defined if nozzle throat is located within current segment
• tube wall thickness "*h*" and thermal conductivity	required
• number of tubes	required
• flag of helical tube wall jacket design	required



Tube wall jacket parameters

The program supports the segments with constant number and variable diameter of the tubes. If you need to define the chamber with variable number of tubes, you have to define several segments each with different corresponding parameters.

Typical mistakes during the analysis run:

- inappropriate number or diameter of tubes, or too high coolant mass flow rate lead to high pressure drop; the message "Negative pressure drop" appears;
- due to inappropriate number or diameter of tubes, or too small mass flow rate the program cannot converge the solution.

Possible solution: change one or several relevant parameters or replace the single cooling segment with two or more segments with optimum parameters.

Channel Wall Jacket Design

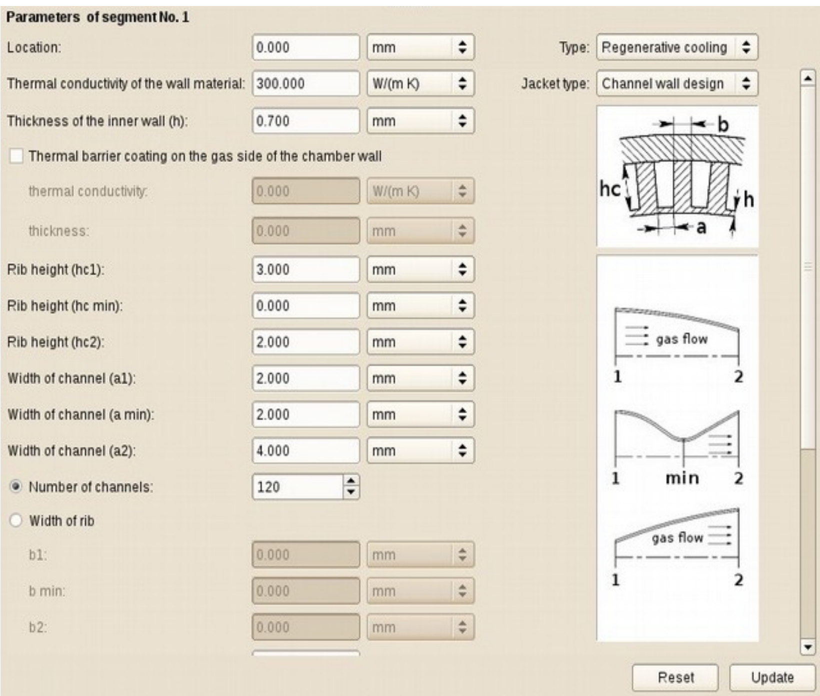
Design-specific configuration of the segment includes the following parameters:

• height of ribs “*hc1*”, “*hc2*” at both sides of the segment	required
• height of ribs “*hc min*” at minimal diameter of the segment	optional; can be defined if nozzle throat is located within current segment
• width of channels “*a1*”, “*a2*” at both sides of the segment	required
• width of channels “*a min*” at minimal diameter of the segment	optional; can be defined

if nozzle throat is located within current segment

- number of channels “*n”* — required
  or
  width of ribs “*b1”*, “*b2”* at both sides of the segment
- width of ribs “*b min”* at minimal diameter of the segment — optional; can be defined if nozzle throat is located within current segment
- angle between channels and generatix — optional

Note: index "1" corresponds to the parameters at the section end which is closer to injector plate; index "2" corresponds to the parameters at the section end which is closer to nozzle exit.



Channel wall jacket parameters

The program supports the segments with constant number of channels and angle between channels and generatix. If you need to define the chamber with variable number of channels and/or angle, you have to define several segments each with different corresponding parameters.

Typical mistakes during analysis run:

- inappropriate geometry and/or number of channels, or too high coolant mass flow rate lead to high pressure drop; the message "Negative pressure drop" appears;

- due to inappropriate geometry and/or number of channels, or too small mass flow rate the program cannot converge the solution.

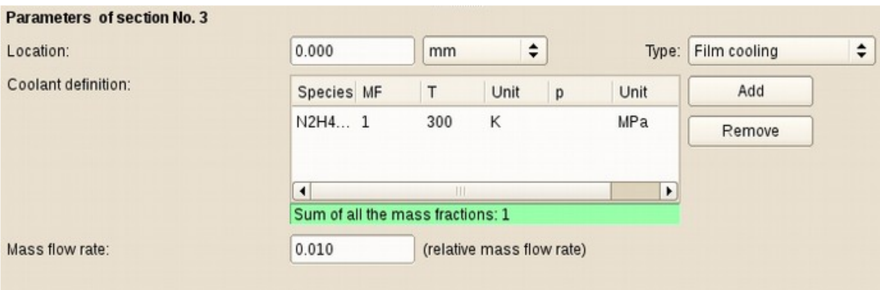
Possible solution: change one or several relevant parameters or replace the single cooling segment with two or more segments with optimum parameters.

Film Cooling

Film cooling provides protection from excessive heat by introducing a thin film of coolant through orifices around the injector periphery or in the chamber wall. This method is typically used in high heat flux regions in combination with regenerative or radiation cooling.

Configuration of the film cooling includes the following parameters:

- location (the distance from the injector plate) required
- coolant definition: required
  component/s,
  initial temperature,
  relative mass flow rate



Film cooling parameters

You can specify either liquid or gaseous species to be used as a film coolant.

Thrust chamber with several cooling segments

You can define more than one cooling segments

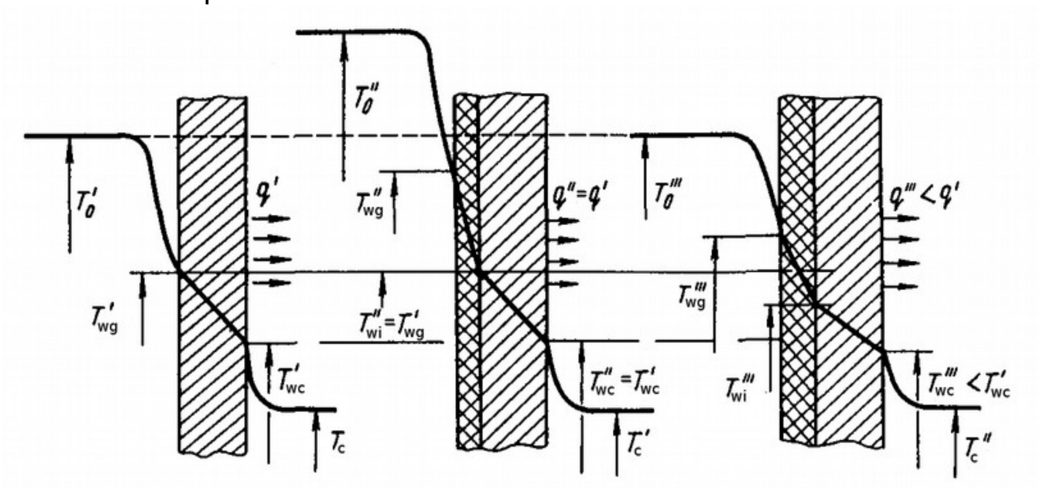
- to combine different types of cooling (e.g. regeneratively cooled combustion chamber + radiatively cooled nozzle extension + one or more film cooling);
- to optimize parameters along the chamber (e.g. number of tubes/channels, tube/channel geometry etc).

Thermal Barrier Coating Layer

The thermal barrier coating (TBC) is used to reduce the temperature gradient across the chamber wall.

The functional principle of a TBC, illustrated in figure below, is to increase the hot gas side wall temperature T_{wg}. This is achieved by applying a ceramic top layer (e.g. Y2O3-stabilized

ZrO2, PYSZ) onto the wall base material. The PYSZ ceramic offers a thermal conductivity λ of about 1.5 W/(m K). In order to prevent a coating overheat (>1500 K) coating thicknesses of less than 0.1 mm are required. These high hot gas side wall temperatures compared to the maximum temperatures of the uncoated wall lead to the remarkable reduction in heat flux.



Functional principle of a TBC

With such a protective layer, the application range of existing regenerative or radiation cooling thrust chambers can be enlarged towards higher combustion chamber pressures or an increased thrust chamber life.

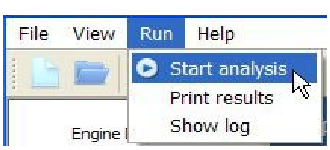
In general a TBC acts as an insulator to the inner liner, reducing the wall heat flux into the wall base material.

If you need to analyze the thermal state of the thrust chambers with TBC consisting of several layers with known thicknesses $\delta_{1,}\delta_{2,}\ldots,\delta_N$ and thermal conductivities $\lambda_{1,}\lambda_{2,}\ldots,\lambda_N$, you may calculate the effective thermal conductivity λ of the TBC with the thickness $\delta_1+\delta_2+\ldots+\delta_N$ from the following relation:

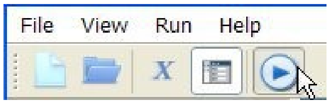
$$\lambda = \frac{\delta_1+\delta_2+\ldots\delta_N}{\frac{\delta_1}{\lambda_1}+\frac{\delta_2}{\lambda_2}+\ldots+\frac{\delta_N}{\lambda_N}}$$

Thermal Analysis

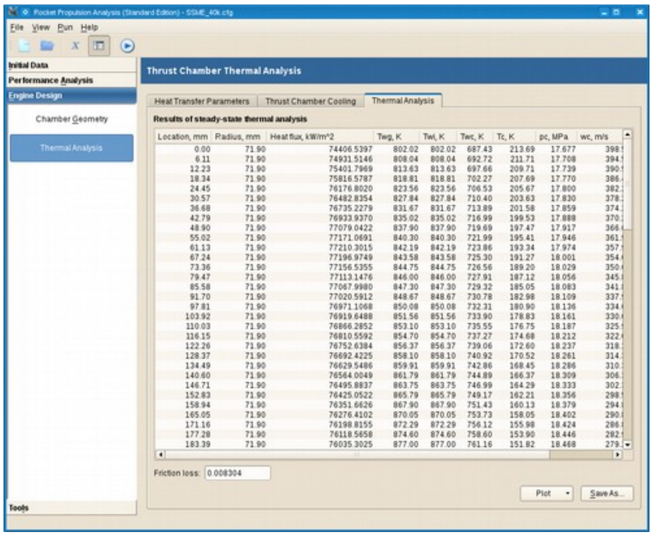
After specifying all required parameters, you can start the analysis by clicking menu **Run**, and then **Start analysis** in menu bar:



or clicking icon **Start** on the main toolbar:



After successful finishing the analysis the program automatically switches to the screen Thermal Analysis, that displays the calculated results.



Thermal analysis results

The thrust chamber geometry is printed out in columns "Location" and "Radius".

The column "Heat flux" displays the calculated heat flux (depending on heat transfer parameters, either the sum of convection heat flux and radiation heat flux or convection heat flux only) at the corresponding location.

The column "T_{wg}" displays the temperature of chamber wall on its hot gas side.

The column "T_{wi}" displays the temperature between the thermal barrier coating layer and chamber wall (if coating is available) or the temperature of chamber wall on its hot gas side (the same as T_{wg}).

The column "T_{wc}" displays the temperature of chamber wall on its cooler side.

The columns "p_c", "T_c", "wc", and "rhoc" display the pressure, temperature, velocity, and density of the coolant correspondingly (if applicable).

Using context menu, you can copy the content of the complete table or single selected row into the clipboard.

Click the button "Save As..." at the right-bottom corner of the screen in order to save the results as ASCII or HTML file.

You can plot the diagrams "Heat Flux vs. Location" or "Temperatures vs. Location", clicking the button Plot at the bottom-right corner of the tab. The location of nozzle throat is shown on the diagram by corresponded vertical marker line.

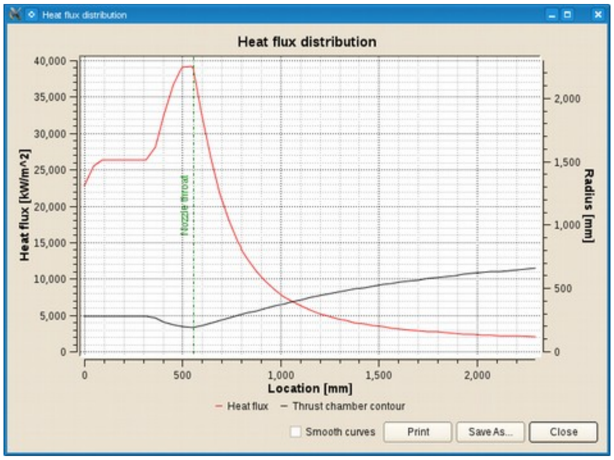


Diagram „Heat flux vs. Location“

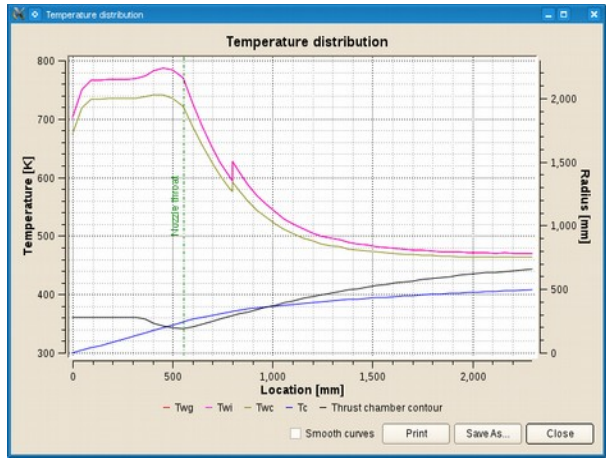


Diagram „Temperatures vs. Location“

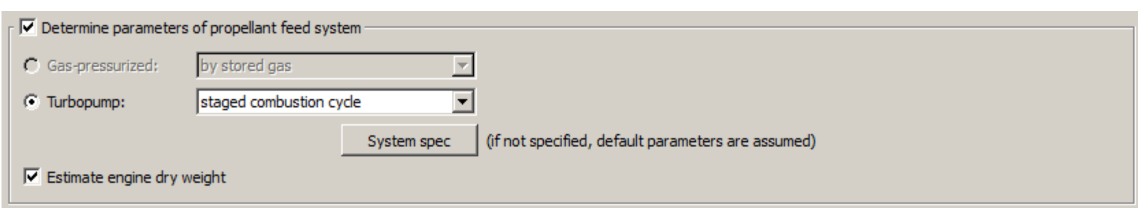
All results are displayed in the current units, defined in the Preferences Dialog.

Propellant Feed System (Cycle Analysis)

The program is capable of performing the liquid-propellant engine cycle analysis.

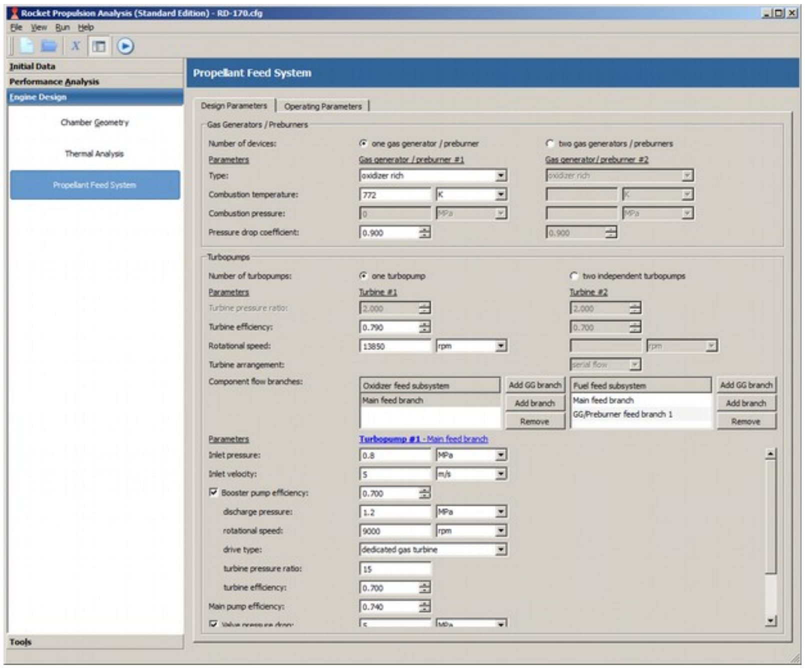
The screen Propellant Feed System consists of 2 tabs *Design Parameters* and *Operating Parameters*.

The input of parameters on the screen *Design Parameters* are only enabled if both flags "Determine thrust chamber size" and "Determine parameters of propellant feed system" on the screen Engine Definition are switched on, and the type of engine cycle is provided:



Design Parameters

On the tab *Design Parameters* user can define design parameters of the propellant feed system.



Propellant feed system design parameters

The program supports analysis of following engine cycles:

- Gas generator cycle
- Staged combustion cycle
- Full flow staged combustion cycle

For selected cycle, the following design parameters can be defined:

• Number of gas generators/preburners: one combustion device two combustion devices	required
• Type of gas generators/preburners: oxidizer-rich fuel-rich In staged combination cycle with two preburners, the type of both combustion devices has to be identical. In full flow staged combination cycle, the type of combustion devices has to be different.	required
• Combustion temperature The value is limited by heat resistance and high-temperature strength of used construction materials. Typical values for oxidizer-rich devices are 650-850 K Typical values for fuel-rich devices are 800-1100 K	required
• Combustion pressure Typical values for gas generator cycle: $p = (0.8 \ldots 0.85)\, p_c$	required for gas generator cycle

For staged combustion cycles the parameter is disabled.	
• Pressure drop coefficient The value defines the pressure drop between injector of the combustion device and turbine inlet. Typical values are 0.9-0.95	required
• Number of independent turbopumps: one turbopump two independent turbopumps If two combustion devices are configured, the only selection possibly is two independent turbopumps	required

For each independent turbopump, the following design parameters can be defined:

• Turbine pressure ratio	required for gas generator cycle
• Turbine efficiency Typical values are 0.5-0.8	required
• Turbine rotational speed Used for estimation of engine dry weight.	required if flag *Estimate engine dry weight* is switched on
• Turbine arrangement: serial flow parallel flow	required for second turbine in configurations with one combustion device
• Main feed branch Defines parameters of component feed branch between engine inlet and • thrust chamber in gas generator cycle • preburner in staged combustion cycles	required
• GG/preburner feed branch Defines parameters of component feed branch between main pump outlet and combustion device.	0 to 2 branches depending on cycle diagram
• Additional branches Defines parameters of additional branch.	optional

For each main feed branch, the following design parameters can be defined:

• Inlet component pressure Typical values are 0.2-0.8 MPa and depend on tank conditions	required
• Inlet component velocity	required

Typical values are 4-10 m/s, for liquid hydrogen 10-20 m/s	
• Main pump efficiency Typical values are 0.6-0.8 For single-shaft turbopumps typical values are: 0.7-0.8 for oxidizer pump 0.55-0.7 for fuel pump	required
• Booster pump efficiency Typical values are 0.5-0.7	optional
• Booster pump discharge pressure	required if booster pump is configured
• Booster pump rotational speed Used for estimation of engine dry weight.	required if flag *Estimate engine dry weight* is switched on and booster pump is configured
• Booster pump drive type: dedicated hydraulic turbine dedicated gas turbine main gas turbine	required if booster pump is configured
• Booster turbine pressure ratio Typical values are 1.1-1.2 for hydraulic turbine, and 5-15 for gas turbine	required if booster pump with dedicated turbine is configured
• Booster turbine efficiency Typical values are 0.5-0.7	required if booster pump with dedicated turbine is configured
• Valve pressure drop Typical values: $\Delta p = (0.1...0.2)p_c$	optional
• Cooling jacket pressure drop Typical values: $\Delta p = (0.25...0.35)p_c$, where lower coefficient is selected for $p_c<6\,MPa$ and upper coefficient is selected for $p_c>8\,MPa$	optional
• Cooling jacket temperature raise	optional (not used in current version)
• Injector pressure drop Typical values: $\Delta p = (0.4...0.8)\sqrt{p_c}$ or $\Delta p = 0.3...1.5\,MPa$	optional

For each gas generator/preburner feed branch, the following design parameters can be defined:

• Assigned relative mass flow rate	Optional (not used in current version)
• Inlet relative cross-section area. Defines the inlet cross-section area of the branch relative to the inlet cross-section area the main branch. Typical values are 0.1-0.3	required
• Kick pump efficiency Typical values are 0.3-0.5 If no kick pump is defined, the pump of the main branch has to produces such a discharge pressure that it fits to inlet of gas generator/preburner. Usually no kick pump is required for gas generator cycle, whereas an absence of kick pump in staged combustion cycle leads to significant pressure raise.	optional
• Valve pressure drop Typical values: $\Delta p = (0.1...0.15) p_c$	optional
• Cooling jacket pressure drop Defines pressure drop in cooling jackets of additional devices (e.g. gas ducts, preburners, etc.) except cooling jackets of the thrust chambers.	optional
• Fixed pressure drop Defines additional pressure drops between main pump outlet and combustion device except pressure drops in valve, cooling jacket and injector.	optional
• Injector pressure drop Typical values: $\Delta p = (0.4...0.8)\sqrt{p_c}$	optional

For each optional feed branch, the following design parameters can be defined:

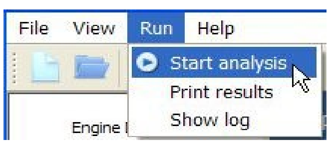
• Name	required
• Relative mass flow rate Defines the mass flow rate through the branch relative to the total mass flow rate at the inlet of the main branch.	required
• Inlet relative cross-section area Defines the inlet cross-section area of the branch relative to the inlet cross-section area the main branch. Typical values are 0.1-0.3	required
• Kick pump efficiency Typical values are 0.3-0.5 If no kick pump is defined, the pump of the main branch has to produces such a discharge pressure that it fits to inlet of gas generator/preburner. Usually no kick pump is required for	optional

gas generator cycle, whereas an absence of kick pump in staged combustion cycle leads to significant pressure raise.	
• Valve pressure drop Typical values: $\Delta p = (0.1...0.15) p_c$	optional
• Cooling jacket pressure drop. Defines pressure drop in cooling jackets. If the branch is used to feed the coolant for the thrust chambers, the typical values: $\Delta p = (0.25...0.35) p_c$, where lower coefficient is selected for $p_c < 6\,MPa$ and upper coefficient is selected for $p_c > 8\,MPa$	optional
• Fixed pressure drop Defines additional pressure drops between main pump outlet and combustion device except pressure drops in valve and cooling jacket.	optional
• Discharge pressure Defines discharge pressure from the branch.	required

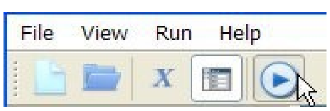
Subsystems of Propellant Feed System

Running Cycle Analysis

After specifying all required parameters, you can start the analysis by clicking menu **Run**, and then **Start analysis** in menu bar:

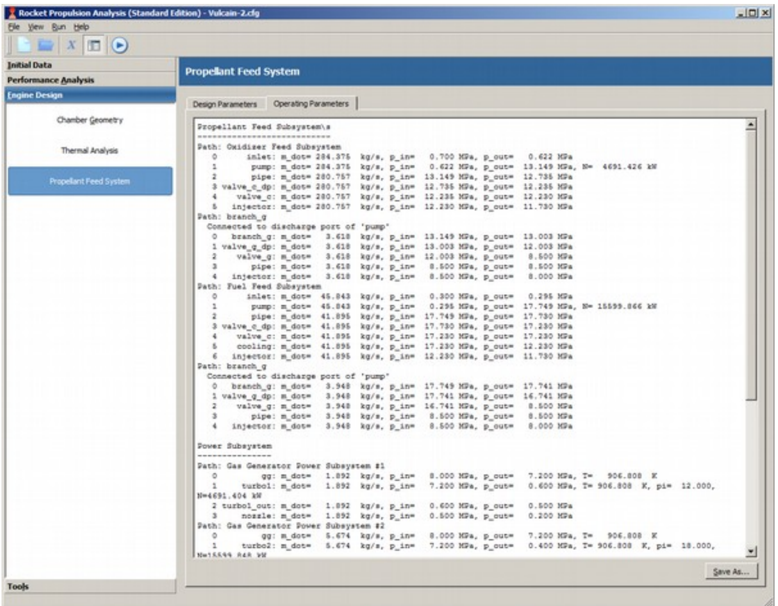


or clicking icon **Start** on the main toolbar:



Operating Parameters

After successful finishing the analysis the program automatically switches to the screen Operating Parameters, that displays the calculated results.



Cycle analysis results

Cycle Performance

Estimated engine performance is printed out below the operating parameters of the feed system:

```
Engine performance
------------------
Chamber performance
   specific impulse (vac):   429.51 s
   specific impulse (opt):   412.88 s
Engine performance
   specific impulse (vac):   419.66 s
   specific impulse (opt):   403.03 s
        correction factor:     0.98
```

For staged combustion cycle, the engine performance is identical to chamber performance.

Engine Dry Weight

If the flag *Estimate engine dry* weight was switched on, the estimated engine weight is printed out at the bottom of the screen *Operating Parameters*:

```
Engine dry weight estimation
----------------------------
                chamber/s:    736.90 kg
              turbopump/s:    457.90 kg
      booster turbopump/s:      0.00 kg
         other components:    625.29 kg
                    total:   1820.08 kg
```

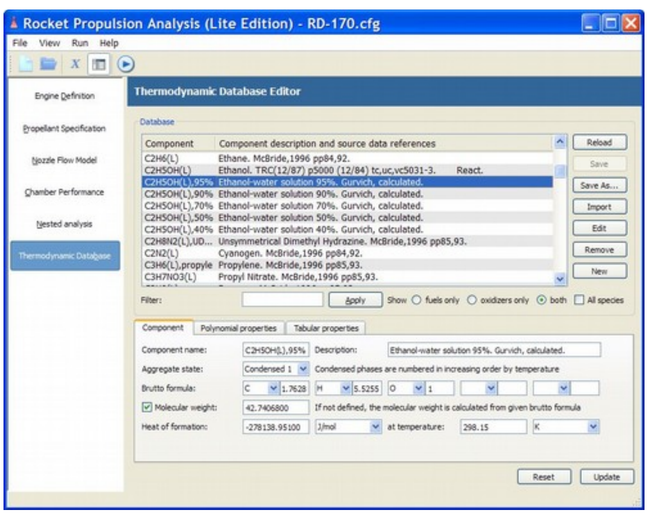
Thermodynamic Database Editor

Thermodynamic Database Editor is an embedded species viewer/editor. Using the tool, you can easily define new propellant components, or import components from *PROPEP* or *CEA2* species databases.

RPA distribution packages contain two database files `resources/thermo.inp` and `resources/properties.inp`. The file `resources/thermo.inp` contains the thermodynamic properties in format described in reference http://www.grc.nasa.gov/WWW/CEAWeb/def_formats.htm.

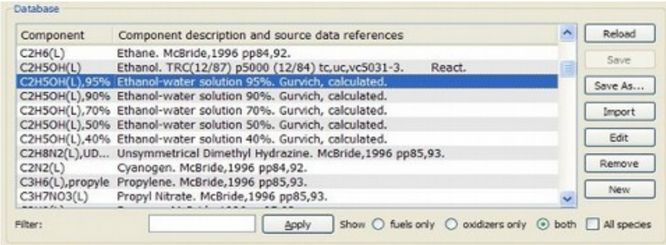
You can define your own species and save it into two additional database files `resources/usr_thermo.inp` and `resources/usr_properties.inp`. These files are not shipped within standard RPA distribution packages, and will not be rewritten after program update.

The tool consists of database viewer at the top of the screen, and species editor at the bottom. You can shrink or heighten the species viewer while the species editor will be heightened or shrunk, dragging the horizontal bar between the viewer and the editor down or up.



Thermodynamic Database Editor

The viewer features the species table, filter and the control buttons. Species table displays currently available species with respect to the filter settings:



You can force the viewer to display the fuels only, or oxidizers only, or both fuels and oxidizers, marking corresponding radio buttons. Mark the check box *"All species"* if you want to see all species, including atomized and/or ionized products of reaction, or keep it unmarked if you want to see only possible propellant components.

The filter pattern is applied to both columns of the table.

The control buttons can be used as follows:

- Click the button **Reload** to reload the default database. Any changes made since the database was last saved will be lost.
- Click the button **Save** to save the changes made since the database was last saved. The program creates a backup-copy of previous version of the database files, adding the character "~" to the name of the file.
- Click the button **Save As...** to save the current database in specified location.
- Click the button **Import** to import the species from external database file in *CEA2* or *PROPEP* formats. After successful loading the external database, the program displays the list of available species is the dialog window. Select one or more species that you want to import and click the button **OK**. All imported species are immediately available for thermodynamic analysis.
- Click the button **Edit** to load the data of the selected species into the species editor. You can also double click the species on the list to load it into the editor.
- Click the button **Remove** to remove the species from the current database. Note that removed species are immediately unavailable for thermodynamic analysis.
- Click the button **New** to reset the editor for creating a new species.

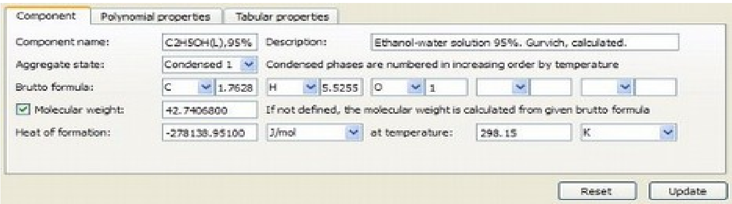
Note: all new species are saved into the user-defined database files `resources/usr_thermo.inp` and `resources/usr_properties.inp`.

Note: although you can import any component from *PROPEP* specues database, do not replace all components already available in *CEA2/RPA* database: the sources of the data in *CEA2* file are NASA Glen thermodynamic database and Gurvich thermodynamic database, both known for their high accuracy.

Note: since *PROPEP* library does not contain the component's temperature, always check standard temperature and tabular data for imported components.

Note: always check the log (click item **Run** in main menu, and then **Show log**; check the tabs *"Warnings"* and/or *"Errors"*) just after the import from *PROPEP* library.

The editor consists of three tabs *Component*, *Polynomial properties*, and *Tabular properties*, and the control buttons at the bottom of the editor:



To save the changed in existing species or save new species, click the button **Update**. To reset the species data, click the button **Reset**.

The tab *Component* displays the information about component, its aggregate state, chemical formula, molecular weight, heat of formation, and the temperature the heat of formation is defined for.

The component name is also an identifier of the species and must be unique within the database. The suffix (L) can be added to the end of the name for the liquid components.

The description usually contains common name of the species, as well as the reference information.

The chemical formula is given as a molecular formula (if applicable), or an exploded formula, followed by its molecular weight. The heat of formation (enthalpy) can be given in one one the units: `J/mol`, `cal/mol`, `kJ/kg`, `kcal/kg`, `Btu/lbm`, `kcal/lbm`. The heat of formation is followed by the temperature (given in one of the units: `K`, `R`, `C`, `F`), for which it has been defined. Note that if polynomial properties are available, the temperature is always `298.15 K` and cannot be changed.

Polynomial properties for the one or more temperature interval are given by 9 coefficients as described in reference http://www.grc.nasa.gov/WWW/CEAWeb/def_formats.htm. Click the button **Add** to add new temperature interval; click the button **Remove** to delete selected temperature interval.

Tabular properties for the one or more pressure and temperature intervals are given by values of specific heat `Cp` (`kJ/mol-K`), density `rho` (kg/m^3) and dynamic viscosity `mu` (`muPa-s`) for each unique combination of pressure and temperature. Click the button **Add p** to add new pressure interval; click the button **Remove p** to delete selected pressure interval. Click the button **Add T** to add new temperature interval; click the button **Remove T** to delete selected temperature interval.

> **Note:** For the components which are supplied together with thermodynamic properties in the polynomial form, you do not need to define the specific heat (define "0" instead).
>
> **Note:** For the gaseous components you do not need to define the density.

In the database file, the tabular data are formatted as follows:

```
!p, MPa      T, K        Cp, kJ/mol-K  rho, kg/m3    mu, muPa-s
Comp_name        2,3
p1          T1          Cp11         rho11        mu11
p1          T2          Cp12         rho12        mu12
```

```
p1        T3        Cp13        rho13       mu13
p2        T1        Cp21        rho21       mu21
p2        T2        Cp22        rho22       mu22
p2        T3        Cp23        rho23       mu23
```

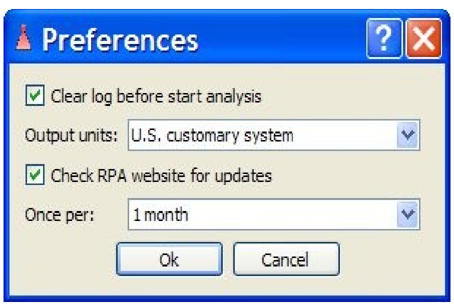
The minimalist data for the component consists of at least two rows:

```
!p, MPa      T, K        Cp, kJ/mol-K  rho, kg/m3    mu, muPa-s
Comp_name        1,2
 0.101325    273.15      0.078        823.0
 0.101325    373.15      0.078        823.0
```

This data defines the constant specific heat `Cp` and constant density rho, and allows to specify the initial temperature in the range `273.15-373.15 K` as well as the initial pressure in the range 0...(the-max-pressure-you-need). Viscosity is not defined (as it will only be required by RPA Standard Edition) and assumed to be equal 0.

Preferences

Dialog window Preferences can be used to set up global configuration parameters. Click the item *Help*, and then *Preferences* in main bar to open the window:



Dialog window "Preferences"

Mark the check box *"Clear log before start analysis"* to force the cleaning the analysis log before each run, or leave it unmarked to let the log accumulate the messages from all runs.

You can define the default output units that will be used by the program to display the results of analysis, selecting either Metric system (SI) or U.S. Customary system on the list Output units.

If selected Metric system (SI), the following units will be used:

Parameter	**Unit**
Temperature	`K (of reaction products)`
Temperature	`K (of propellant components)`
Pressure	`MPa`

Parameter	Unit
Specific impulse	m/s
Velocity	m/s
Mass flow rate	kg/s
Mass flux	kg/m^2-s
Thrust	N, kN
Density	kg/m^3
Enthalpy	kJ/kg, J/mol
Entropy, Specific heat, Gas constant	kJ/kg-K, J/mol-K

If selected U.S. Customary system, the following units will be used:

Parameter	Unit
Temperature	F (of reaction products)
Temperature	R (of propellant components)
Pressure	psi
Specific impulse	ft/s
Velocity	ft/s
Mass flow rate	lbm/s
Mass flux	lbm/ft^2-s
Thrust	lbf
Density	lbm/ft^3
Enthalpy	Btu/lbm, Btu/lb-mol
Entropy, Specific heat, Gas constant	Btu/lbm-R, Btu/lb-mol-R

Mark the check box *"Check RPA website for updates"* to force the program to check for available updates, or leave it unmarked to disable this function. You can also define how often should the program perform the check, selecting one of the following items on the list *"Once per"*:

- RPA start-up
- 1 day, 1 week
- 1 month

In order to force the program to check for available updates, click the item **Help**, and then **Check for Updates** in main bar.

Input and Output Units

You can enter the values of input parameters in any available units, freely mixing Metric (SI) and U.S. Customary systems. The program automatically converts all entered values to the Metric system, which is standard internal representation of both input parameters and results of calculation.

The following conversion factors are used to convert values in non-SI units to values in SI units:

Name	Value	Description
CONST_ATM	101325.0	Conversion factor from atm to Pa
CONST_AT	98066.5	Conversion factor from at (technical atmosphere) to Pa
CONST_BAR	100000.0	Conversion factor from bar to Pa
CONST_PSI	6894.75729316836	Conversion factor from psi (pound-force per square inch) to Pa
CONST_T0	298.15 K	Temperature 25 C
CONST_R0	8.314472 J/(mol·K)	Universal Gas Constant
CONST_G	9.80665 m/s^2	
CONST_POUND	0.45359237	Conversion factor from lbm (pound mass) to kg
CONST_POUND_FORCE	(CONST_POUND*CONST_G)	Conversion factor from lbf (pound-force) to N (newton)
CONST_FOOT	0.3048	Conversion factor from international foot to m
CONST_INCH	0.0254	Conversion factor from inch to m
CONST_MILE	(5280.0*CONST_FOOT)	Conversion factor from international mile to m
CONST_LBM_FOOT3	(CONST_POUND/CONST_FOOT^3)	Conversion factor from "lbm/ft^3" to "kg/m^3
CONST_LBM_INCH3	(CONST_POUND/CONST_INCH^3)	Conversion factor from "lbm/inch^3" to "kg/m^3"
CONST_MASS_FLUX	(CONST_POUND/CONST_FOOT^2)	Conversion factor from "lbm/(ft^2·s)" to "kg/(m^2·s)"
CONST_LBM_MOLE	(1000.*CONST_LBM)	Conversion factor from lb-mole to mole
CONST_BTU	1055.05585262	Conversion factor from Btu to J
CONST_CAL	4.1868	Conversion factor from calorie to J
CONST_BTU_LBM	(CONST_BTU/CONST_LBM)	Conversion factor from Btu/lbm to J/kg
CONST_BTU_LBM_MOLE	(CONST_BTU/CONST_LBM_MOLE)	Conversion factor from Btu/lb-mol to J/mol

Name	Value	Description
CONST_BTU_LBM_R	(1000.*CONST_CAL)	Conversion factor from Btu/(lbm·R) to J/(kg·K)
CONST_BTU_LBM_F	CONST_BTU_LBM_R	Conversion factor from Btu/(lbm·F) to J/(kg·K)
CONST_BTU_LBM_MOLE_R	CONST_BTU_LBM_R	Conversion factor from Btu/(lb-mol·R) to J/(mol·K)

The Metric system is also used by default to display the results of calculation. You can change it to U.S. Customary system, using dialog window Preferences.

References:

- SP-811. NIST Guide for the Use of the International System of Units (SI). B.8 Factors for Units Listed Alphabetically
- Glossary of terms and table of conversion factors used in design of chemical propulsion systems. NASA SP-8126. 1979.
- George P. Sutton, Oscar Biblarz. Rocket Propulsion Elements, 7th Edition (pp.727-729).
- NASA-STD-3000. Man-Systems Integration Standards. Volume II. Appendix E - Units of Measure and Conversion Factors

Scripting API

The scripting language provided is based on the ECMAScript scripting language, as defined in standard ECMA-262.

Scripting utility implements binding to many internal functions of RPA, so that the user can program and run the following tasks:

- load, manipulate and write configuration files
- search the thermodynamic database by species name
- get thermodynamic properties of the species
- prepare mono- bi- and multi-propellant compositions
- run typical combustion problems `(p,H)=const`, `(p,S)=const`, `(p,T)=const`
- run typical rocket propulsion problems
- use in a custom JavaScript program all features listed above

Scripting utility can be started in either an interactive mode or a batch mode.

API Reference

Besides standard ECMA-262 API, RPA Scripting Utility provides the domain-specific API. Reference documentation is available online at http://www.rocket-propulsion.com/downloads/2/docs/scripting_api/index.htm