Imperial College London

Open Source Combustion Instability Low Order Simulator for Longitudinal Modes

(OSCILOS_long)

User guide for version 1.4

Developed by Dr. Jingxuan Li, Dong Yang, Charles Luzzato
and Dr. Aimee S. Morgans*

Programmed with MATLAB 2014a
First released on April 23, 2014
Website: http://www.oscilos.com/
Contact: jingxuan.li@imperial.ac.uk and a.morgans@imperial.ac.uk

What is OSCILOS?

Imperial College London

- The open source combustion instability low-order simulator (OSCILOS) is an open source code for simulating combustion instability. It is written in Matlab®/ Simulink® and is very straightforward to run and edit.
- It can simulate both longitudinal and annular combustor geometries. It represents a combustor as a network of connected modules.
- The acoustic waves are modeled as either 1-D plane waves (longitudinal combustors) or 2-D plane/circumferential waves (annular combustors).
- A variety of inlet and exit acoustic boundary conditions are possible, including open, closed, choked and user defined boundary conditions.
- The response of the flame to acoustic waves is captured via a flame model; flame models ranging from linear n-τ models to non-linear flame describing functions, either defined analytically or loaded from experimental / CFD data, can be prescribed.
- The mean flow is calculated simply by assuming 1-D flow conditions, with changes only across module interfaces or flames.
- This current version is for longitudinal modes. This assumes a longitudinal/cannular/can combustor geometry, or an annular geometry but where only plane acoustic waves are known to be of interest.

Who is developing OSCILOS?

Imperial College London

- OSCILOS is being developed by Dr Aimee Morgans, Dr. Jingxuan Li, Dong Yang and co-workers in the Department of Aeronautics, Imperial College London, UK.
- More details about the development team are available on the website:

  http://www.oscilos.com/

- Current team members:

  Dr. Aimee S. Morgans (project lead)

  Dr. Jingxuan Li

  Dong Yang

  Charles Luzzato

  Dr. Xingsi Han

- The latest version of OSCILOS is available from our Github repository:

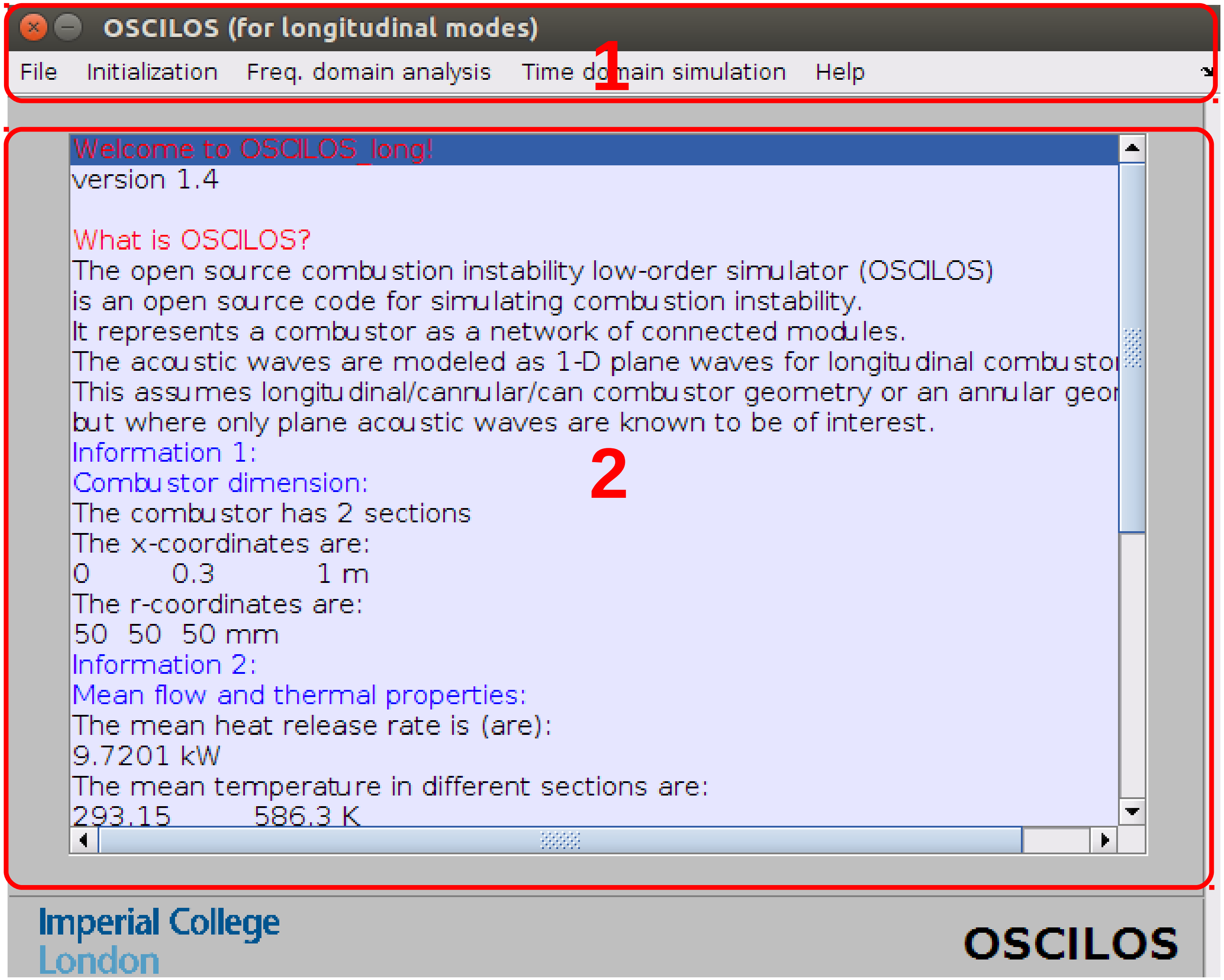
  https://github.com/MorgansLab/

  Contributions are welcome and can be submitted with GitHub pull request. These will be reviewed and accepted by the team.

Required Matlab toolboxes

- Control System Toolbox
- Matlab
- Optimization Toolbox
- Robust Control Toolbox
- Simulink
- Symbolic Math Toolbox

Main console



1. Menu Bar

The menu bar organizes the GUI menu hierarchy using a set of pull-down menus.

A pull-down menu contains items that perform executed actions.

2. Information window

Key information from the program run is printed in the information window.

Menu Bar

Imperial College London

- **File**
  - New case
  - Load...
  - Save...
- **Initialization**
  - Chamber dimensions
  - Thermal properties
  - Flame model
  - Boundary conditions
- **Frequency domain analysis**
  - Eigenmode calculation
- **Time domain simulation**
  - Examination of the Green's function
  - Parameter configuration
  - simulation
  - Results output and plot
- **Help**
  - About
  - User guide

File

New case: This menu option is used to clear current results and create a new calculation.

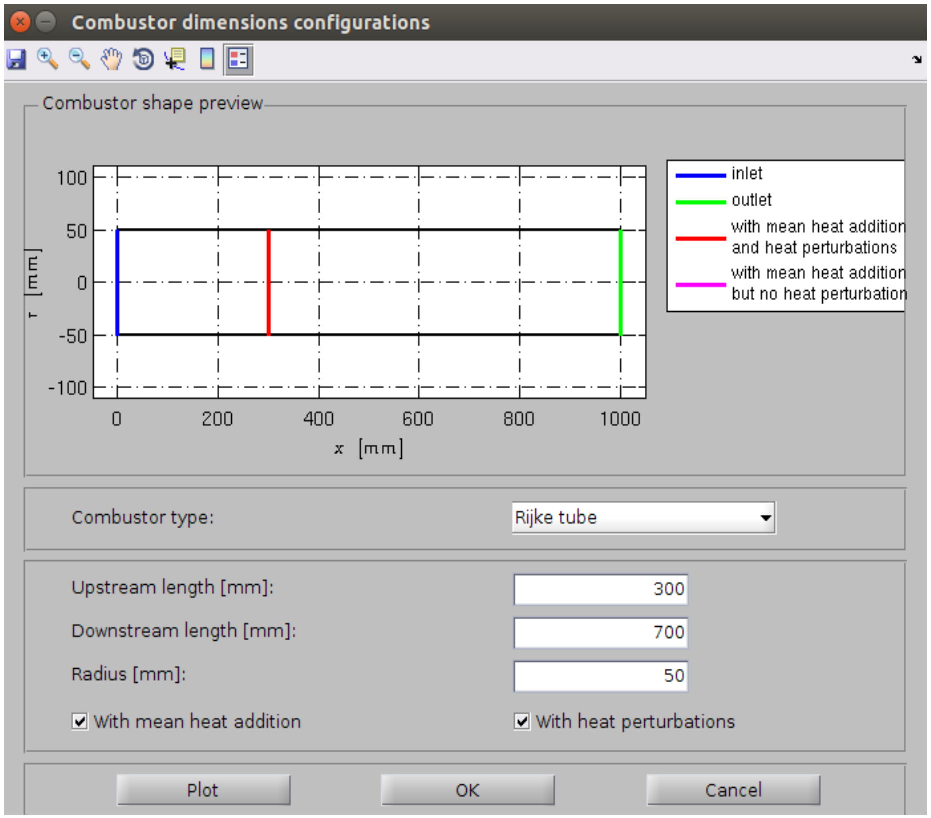
Load...: This menu option is used to load data from existing Mat file. The current calculation results will be deleted once the user clicks Yes.

Save...: This menu option is used to save the current calculation results as a Mat file.

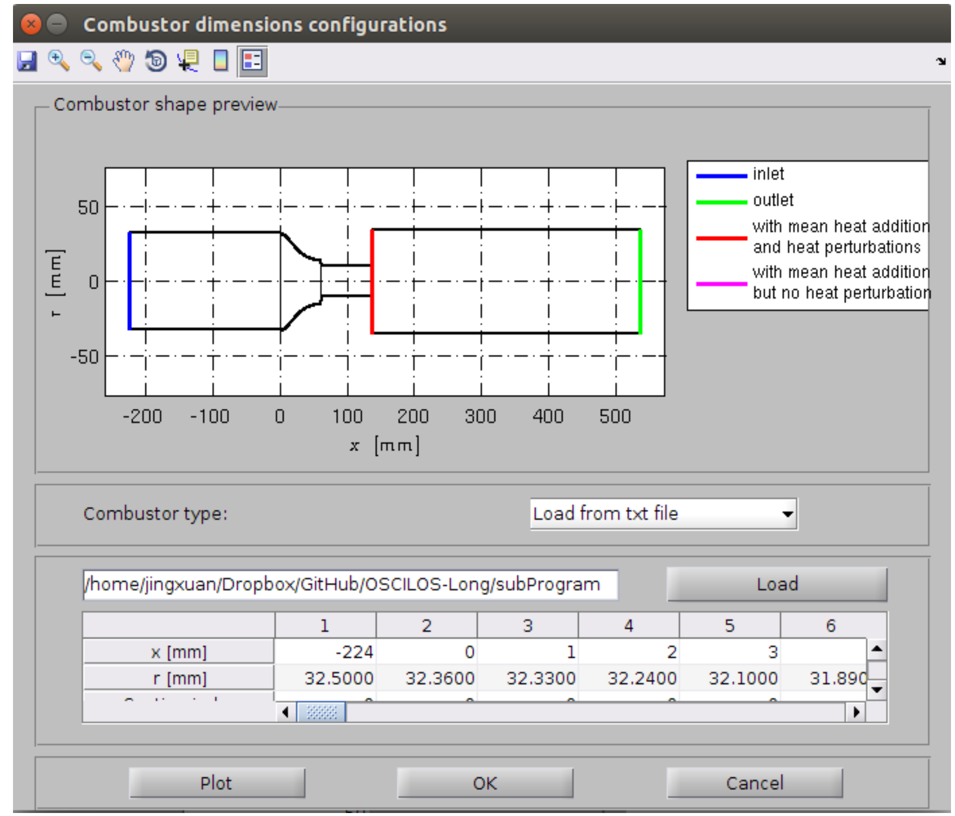
Initialization / Chamber dimensions

Imperial College London

This menu option is used to set the geometric dimensions of the combustor.



(a) Interface for Rijke tube



(b) Combustor geometry is loaded from external txt file

Initialization / Chamber dimensions

Imperial College London

- The length and radius of the combustor can be quickly configured for the case of a Rijke tube.
- For complicated combustor geometries, users can load the information from an external txt file by clicking load. It is better to create the txt file as the form of CD_example.txt in the subProgram folder. The format of data is shown in the following table:

x [m]	r [m]	Interface index [-]	Module index [-]
-0.830	0.017	0	0
-0.750	0.017	0	1
-0.720	0.050	0	0
-0.520	0.050	0	1
-0.450	0.017	0	0
0	0.035	11	0
0.350	0.035	0	0

where,

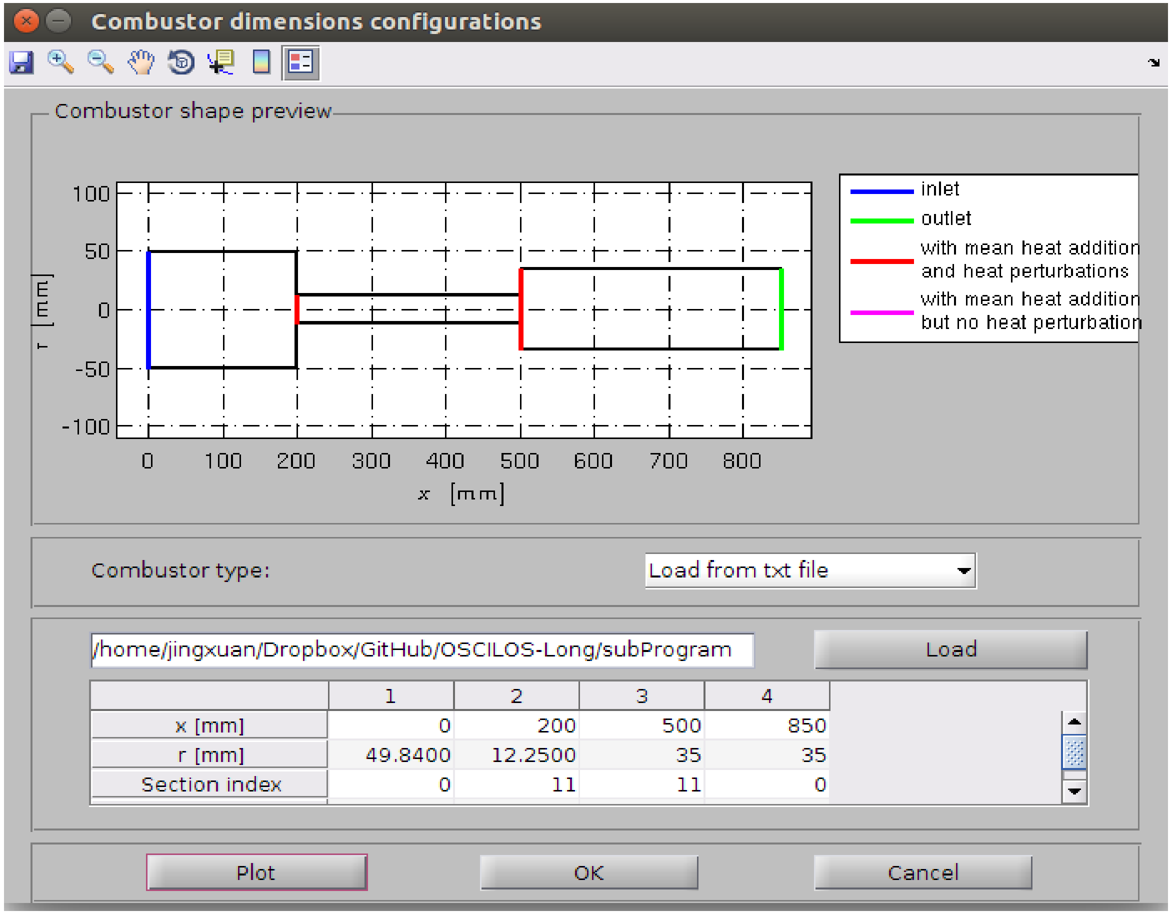
- ✓ x means axial position of each sectional interface;
- ✓ r indicates the radius of each section;
- ✓ Interface index represents the type of interface between modules: '0': a simple area change; '10': with heat addition and without heat perturbation; '11': with heat perturbations.
- ✓ Module index indicates the type of tube between this and the following interface: '0': straight constant area duct; '1': duct with linearly changing radius.

- The schematic of the combustor can be previewed by clicking Plot.
- The current configuration is saved by clicking OK. The key information will be printed in the information window.

Initialization / Chamber dimensions

Imperial College London

OSCILOS can account for cases of multiple heat sources (multi-flames), by setting the section indices to '10' or '11' for the desired interfaces. The following figure shows the schematic view of an example with two flames, in which the red lines are used to represent the unsteady heat sources (flames).

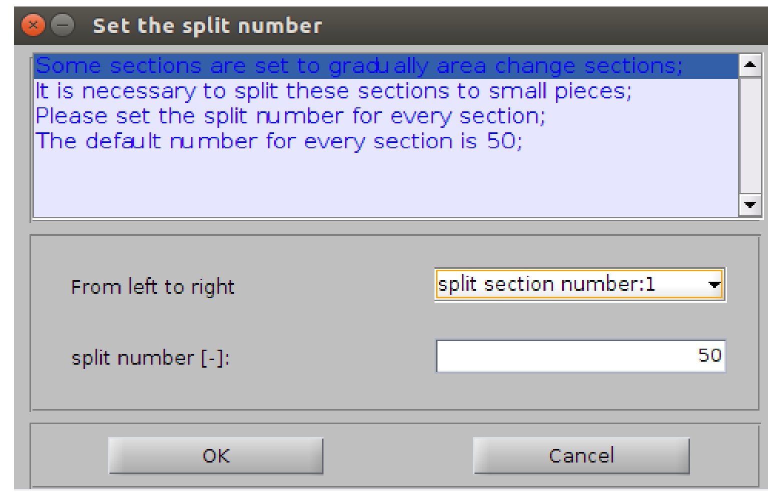


Initialization / Chamber dimensions

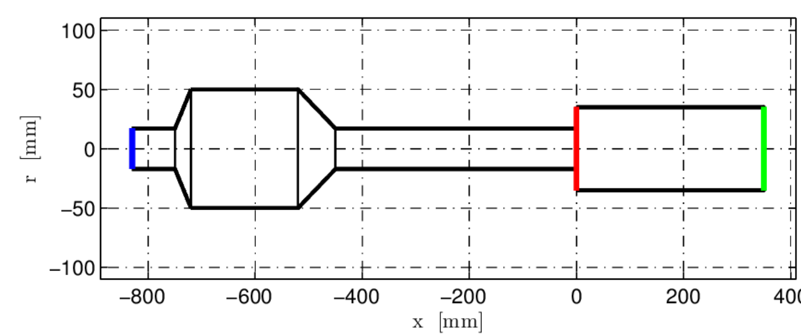
Imperial College London

- OSCILOS can account for geometry modules whose sectional radius gradually changes with axial location.
- For example, the following table shows the combustor dimensions of the key interfaces. The module index at interface 2 is "1", meaning that the radius of the section between interfaces 2 and 3 changes linearly with axial position.

x [m]	r [m]	Interface index [-]	Module index [-]
-0.830	0.017	0	0
-0.750	0.017	0	1
-0.720	0.050	0	0
-0.520	0.050	0	1
-0.450	0.017	0	0
0	0.035	11	0
0.350	0.035	0	0



- This kind of module is split into equally spaced segments for numerical treatment. The number of splits or segments can be set in a jumped window as shown in the top right figure.
- The schematic view of the above combustor is shown in the bottom right figure.

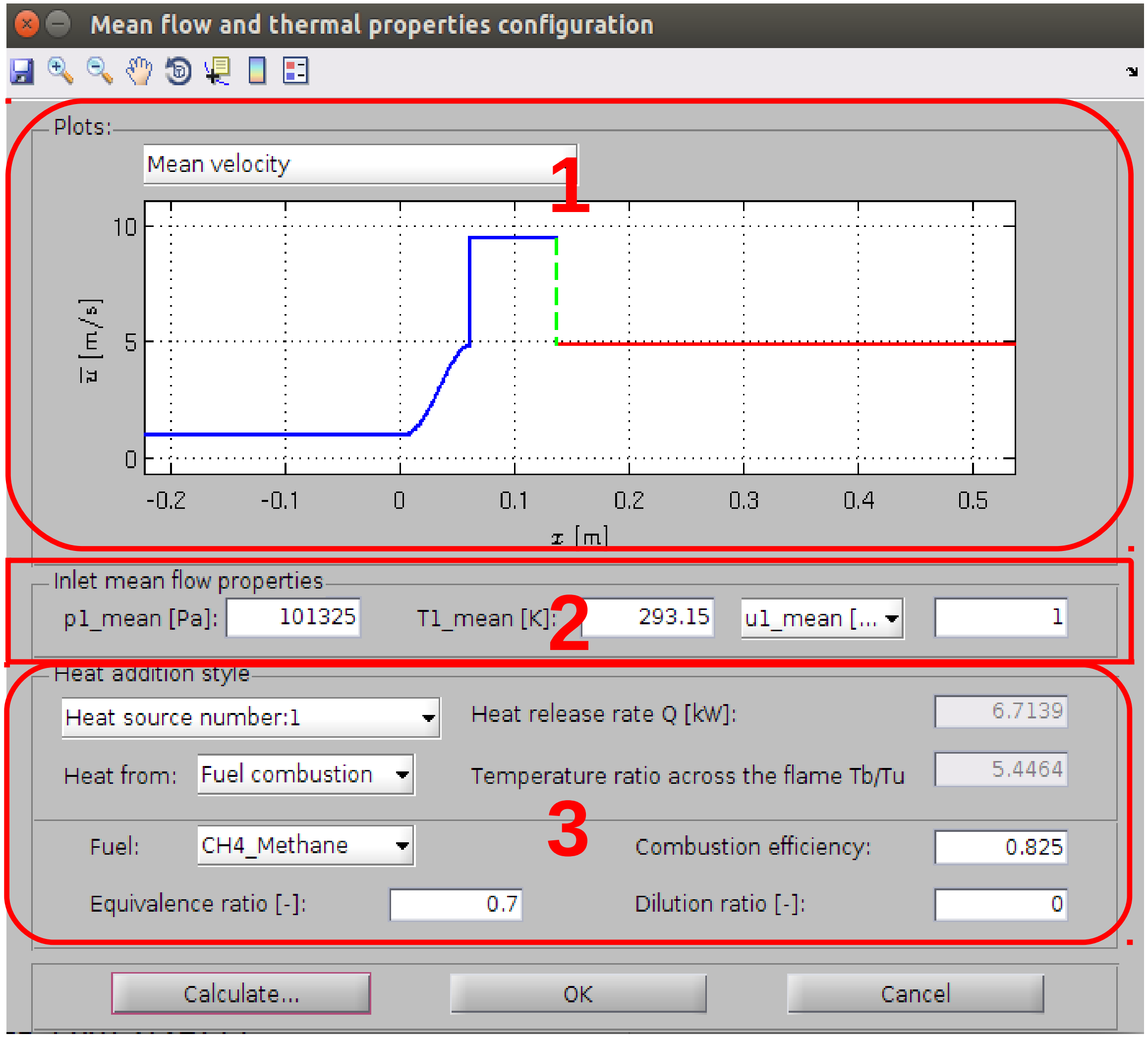


Jingxuan LI
Department of Aeronautics

Initialization / Thermal properties

Imperial College London

This menu is used to set the inlet mean flow properties and the mean heat addition.



1. Plot panel

This panel is used to plot the distribution of mean velocity and mean temperature with axial position.

2. Mean flow properties panel

The inlet mean pressure, temperature and flow velocity/Mach number are set in this panel.

3. Mean heat addition panel

This panel is used to configure the heat addition, once there is (are) heat source(s) inside the combustor.

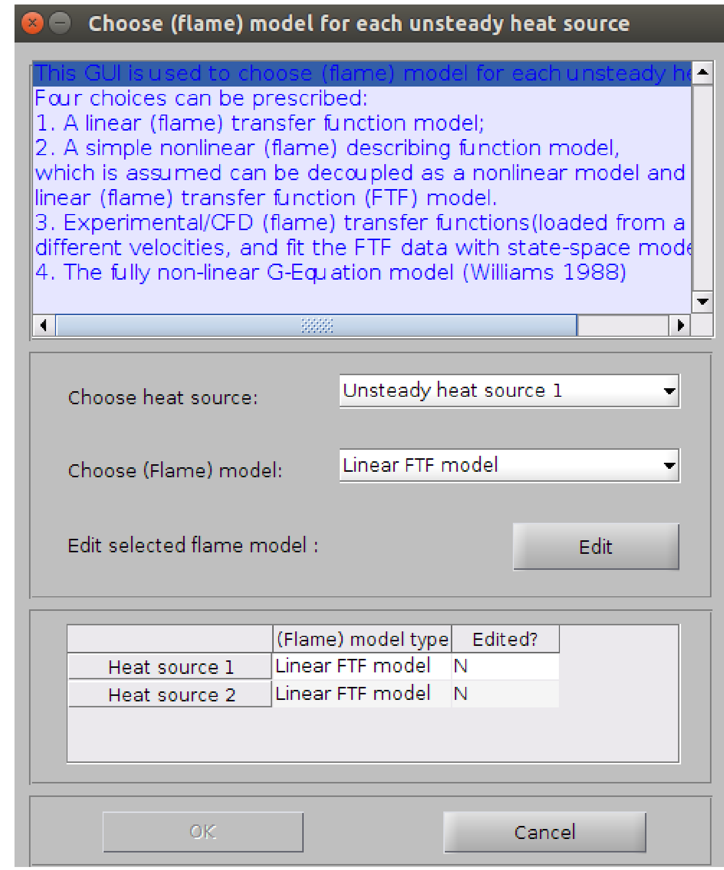
Initialization / Thermal properties

Imperial College London

- The mean flow and thermal properties within each combustor module are considered uniform for constant sectional area modules.
- Modules with varying sectional radius are split into equally spaced segments for numerical treatment. The mean flow and thermal properties are considered uniform in each segment.
- Users can choose a model for the (mean) heat addition from the drop-down menu.
  - ✓ Heat from a heating grid (Tb/Tu is given), where Tu represents the temperature upstream of the grid, and Tb downstream of the grid. This is often used in the case of a Rijke tube.
  - ✓ Heat from fuel combustion.
    1. Users can choose the fuel from the pop-up menu, including CH4, C2H4, C2H6, C3H8, C4H8, C4H10 (n-butane), C4H10 (isobutane) and C12H23 (Jet-A).
    2. Users can also set:
       (a) Equivalence ratio
       (b) Combustion efficiency
       (c) Dilution ratio, which is used when a bias flow is accounted for.
- The mean properties in different modules can be calculated by clicking "Calculate" once all the inputs have been completed.
- For multiple flames cases, users need to set the parameters for each heat source. The corresponding heat release rate for each heat source is shown in the text box after calculation.
- The distribution of mean temperature and mean flow velocity can be previewed by clicking "Plot figure".
- The current configuration is saved by clicking "OK". The key information will be printed in the information window.

Initialization / Flame model

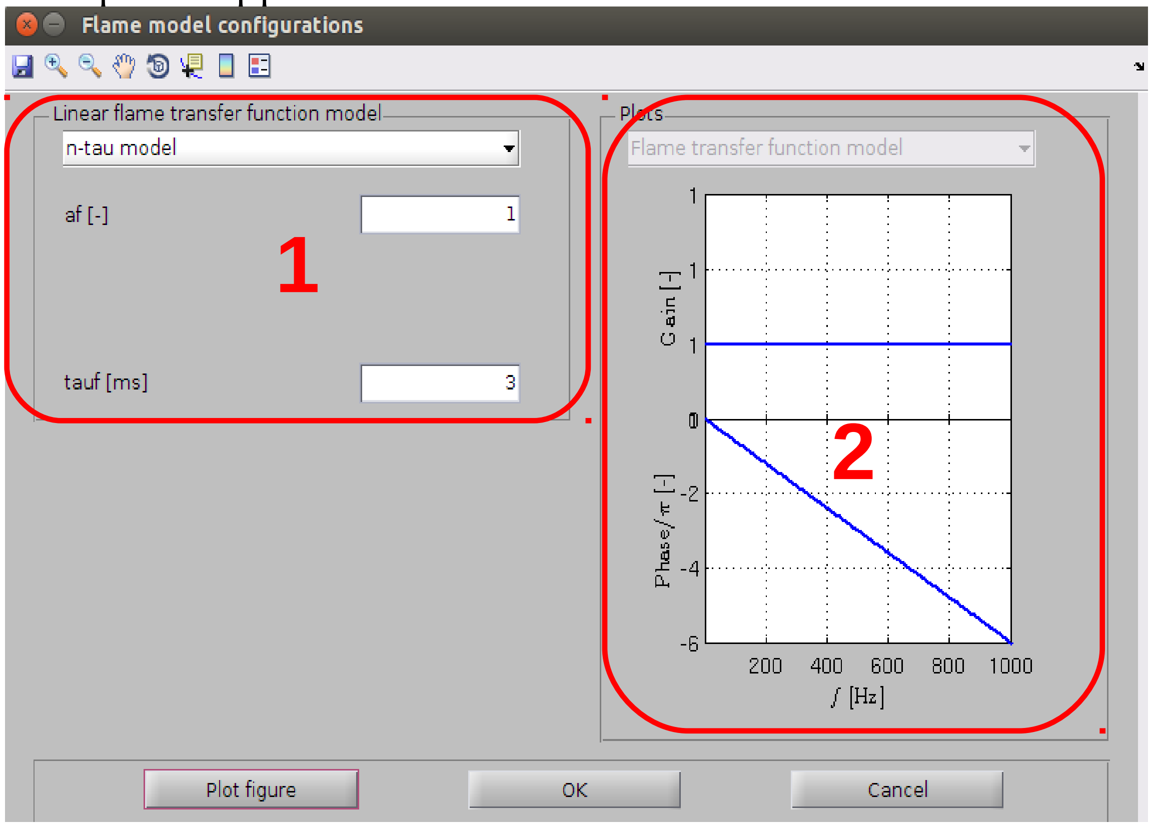
- This panel is used to choose (flame) model for each unsteady heat source.
- The (flame) model describes how the (normalized) unsteady heat release rate of the unsteady heat source (flame) responds to (normalized) velocity fluctuations.
- Four choices can be prescribed:
  - ✓ A linear (flame) transfer function model;
  - ✓ A simple nonlinear (flame) describing function model, which is assumed can be decoupled as a nonlinear saturation model and a linear (flame) transfer function (FTF) model.
  - ✓ Experimental/CFD (flame) transfer functions (loaded from an external mat file) for different velocity perturbations upstream of the flame. The FTF data are fitted within OSCILOS with state-space models.
  - ✓ The fully non-linear G-Equation model (Williams 1988).



- When users choose a heat source and a (flame) model from the drop pop-up menus and click Edit, a corresponding window appears for detailed configuration.
- The button OK will be enable once the flame models for all unsteady heat sources have been configured.

Initialization / Flame model / Linear FTF

This panel appears once users choose linear FTF model.

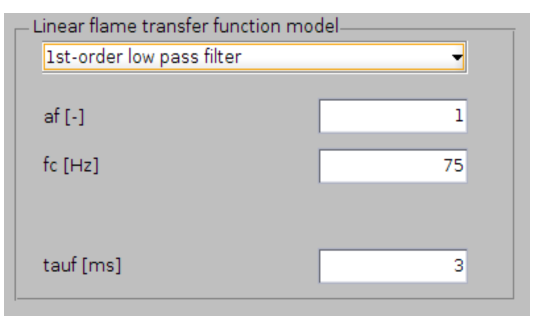


1. Parameters configuration panel

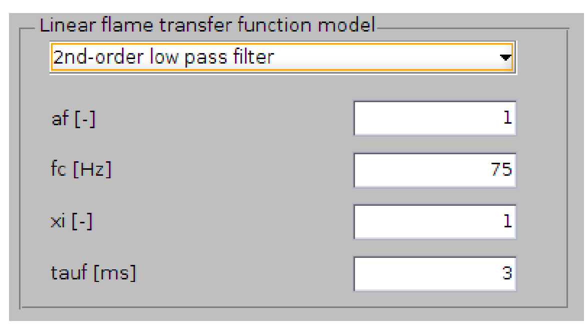
This panel is used to configure the parameters of the flame model. Users can choose the linear FTF from the drop-down menu. The left figure shows the n-tau model configuration panel. Panels corresponding to the other 3 choices are shown in the 3 bottom figures.

2. Plot panel

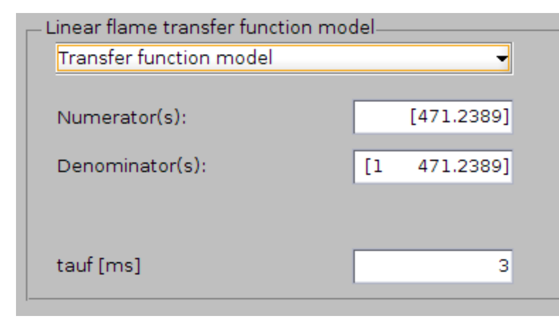
The evolutions of gain and phase lag of the FTF with frequency are plotted once users click Plot figure.



(a) 1st order low pass filter model



(b) 2nd order low pass filter model



(c) polynomial transfer function

Flame transfer function

Imperial College London

Four kinds of flame transfer function models ($\mathcal{T}_u(s) = \hat{\dot{q}}(s)/\bar{\dot{q}} \Big/ \hat{u}(s)/\bar{u}$) can be prescribed analytically: the first three involve:

1. Crocco's famous $n - \tau$ model:

$$\mathcal{T}_u(s) = a_f e^{-\tau_f s}$$

2. the $n - \tau$ model filtered by a first order filter:

$$\mathcal{T}_u(s) = \frac{\omega_c}{s + \omega_c} a_f e^{-\tau_f s}$$

3. the $n - \tau$ model filtered by a second order filter:

$$\mathcal{T}_u(s) = \frac{\omega_c^2}{s^2 + 2\xi\omega_c s + \omega_c^2} a_f e^{-\tau_f s}$$

- s: the Laplace variable
- a_f: gain
- τ_f: time delay
- $f_c = \omega_c \big/ 2\pi$: cut-off frequency
- ξ: damping ratio

The fourth option is a user-defined FTF model using a polynomial transfer function by inputting the numerator coefficients $\mathbf{b}$ and denominator coefficients $\mathbf{a}$. The order of the numerator should not be larger than that of denominator $n \leqslant m$.

$$\mathcal{T}_u(s) = \frac{b_1 s^{n-1} + b_2 s^{n-2} + \cdots + b_{n-1} s + b_n}{a_1 s^{m-1} + a_2 s^{m-2} + \cdots + a_{m-1} s + a_m}$$

Imperial College London

Two kinds of nonlinear flame describing functions can be prescribed. The first is an abrupt heat release rate ratio saturation model proposed by Dowling (JFM:1997), which can be mathematically expressed as:

$$\frac{\dot{q}'}{\bar{\dot{q}}} = \begin{cases} \left(\dfrac{\dot{q}'}{\bar{\dot{q}}}\right)_L & \text{for } \left|\dfrac{\dot{q}'}{\bar{\dot{q}}}\right| \leq \alpha \\ \alpha \ \mathrm{sgn}\left(\dfrac{\dot{q}'}{\bar{\dot{q}}}\right) & \text{else} \end{cases}$$

Where α is a constant associated with the saturation ($0 \leq \alpha \leq 1$) and $\left(\dot{q}' / \bar{\dot{q}}\right)_L$ denotes the heat release rate ratio for weak perturbations, which can be calculated from the linear flame transfer function.

The second nonlinear model was recently proposed by Li and Morgans (JSV:2015). The nonlinear flame describing function depends on s and velocity ratio $\hat{u}_1/\bar{u}_1$ and it is assumed here can be decoupled as:

$$\widetilde{G}(\hat{u}_1/\bar{u}_1, s) = \mathcal{L}(\hat{u}_1/\bar{u}_1)\widetilde{\mathcal{T}}_u(s)$$

where the superscript $\hat{}$ indicates the signal amplitude. The nonlinear function $\mathcal{L}(\hat{u}_1/\bar{u}_1)$ describes the saturation of heat release rate with velocity perturbations $\hat{u}_1/\bar{u}_1$, and $\mathcal{L}(\hat{u}_1/\bar{u}_1) = \widetilde{G}(\hat{u}_1/\bar{u}_1, 0)$. The mathematical link is:

$$\frac{\hat{\dot{q}}(\hat{u}_1/\bar{u}_1, 0)}{\bar{\dot{q}}} = \mathcal{L}(\hat{u}_1/\bar{u}_1)\frac{\hat{u}_1}{\bar{u}_1} = \int_0^{\hat{u}_1/\bar{u}_1} \frac{1}{1 + \left(\xi + \alpha\right)^{\beta}} \mathrm{d}\xi$$

where α and β are two coefficients which determine the shape of the nonlinear model. One may also introduce a simple nonlinear model of the time delay, using the mathematical description:

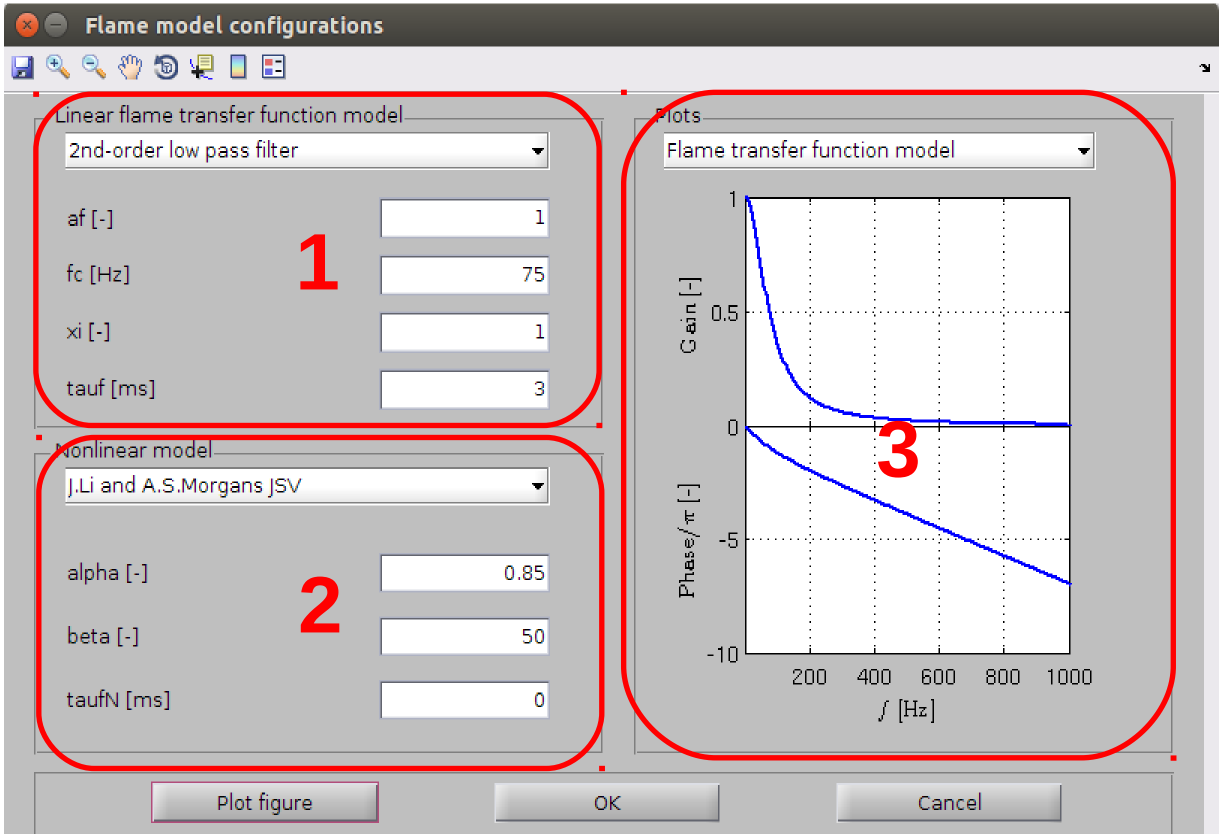
$$\tau_f = \tau_f^0 + \tau_f^N \left(1 - \mathcal{L}(\hat{u}_1/\bar{u}_1)\right)$$

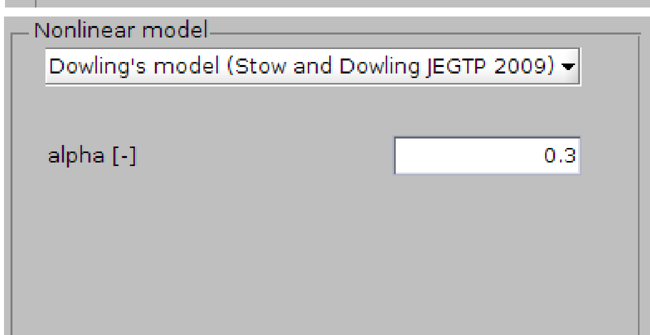
where τ_f^0 means the time delay when $\hat{u}_1/\bar{u}_1 = 0$ and τ_f^N is a time delay to describe the change of τ_f as $\mathcal{L}$ changes.

Initialization / Flame model /Nonlinear FDF model

Imperial College London

This panel appears once users choose nonlinear FDF model.





1. FTF parameters configuration panel

This panel is used to configure the parameters of the linear flame transfer function. Users can choose a linear FTF from the drop-down menu (see slide 15).

2. Nonlinear model parameters configuration panel

This panel is used configure the saturation limit value (alpha) for Dowling's model (as shown in the bottom figure) or parameters for the second nonlinear saturation model.

Imperial College London

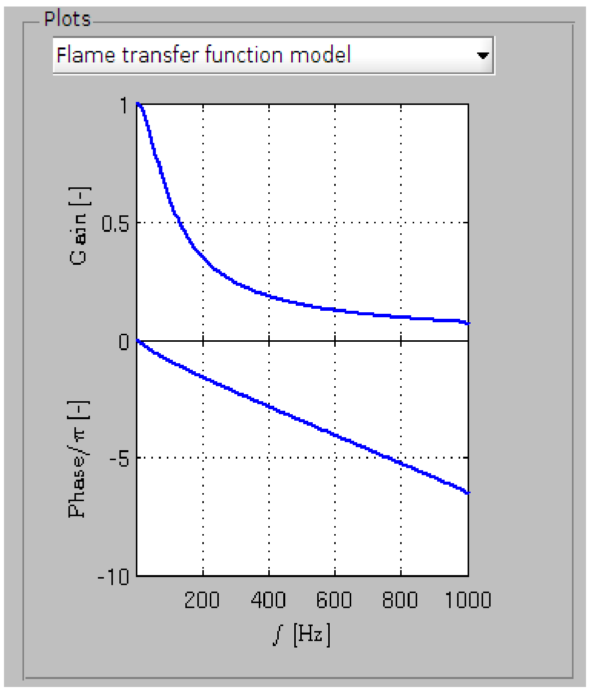
3. Plot panel

Three kinds of plots can be prescribed:

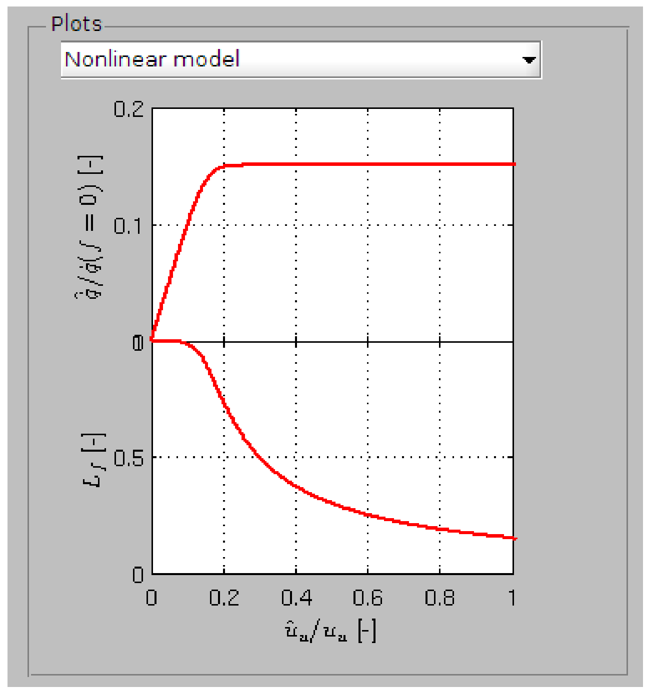
(1) Flame transfer function

(2) Nonlinear saturation model

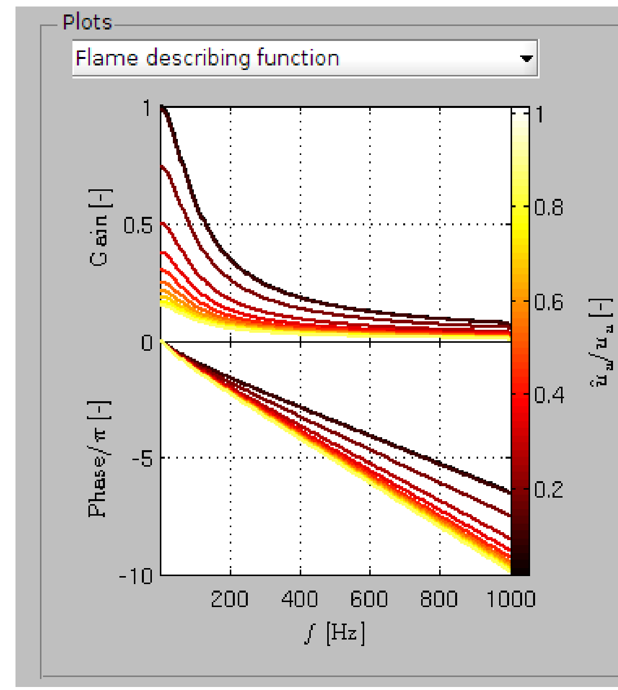
(3) Flame describing function



(a) Flame transfer function



(b) Nonlinear saturation model



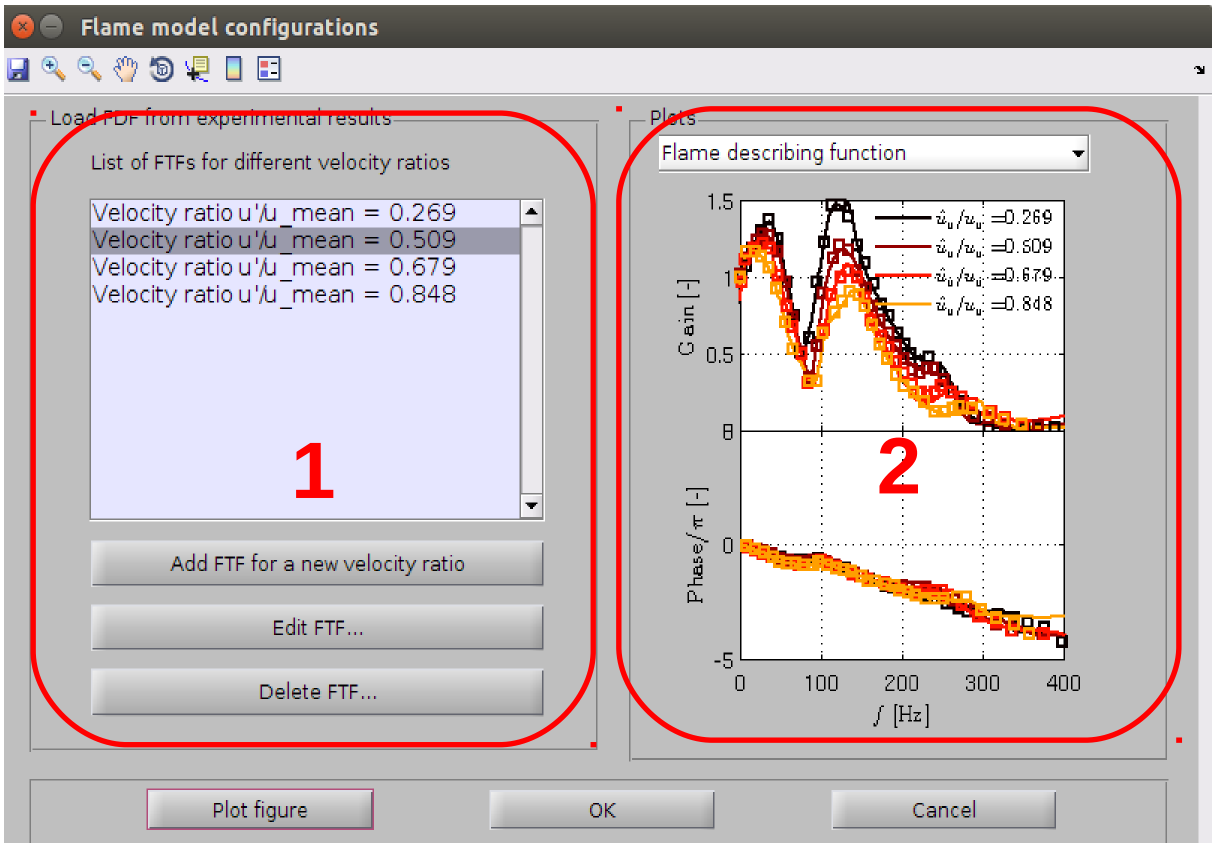
(c) Flame describing function

Jingxuan LI
Department of Aeronautics

Initialization /Flame model /Loaded and fitted FDF from experiment or CFD data

Imperial College London

This panel appears once users choose Experimental/CFD fitted FDF.



1. FDF editing panel

Users can add, edit or remove the flame transfer functions for different flame velocity perturbation levels obtained from experiment or CFD simulation.

2. Plot panel

The original data and fitted FDF can be viewed from this plot.

Initialization /Flame model /Loaded and fitted FDF from experiment or CFD data

Imperial College London

- Once the button Add FTF for a new velocity ratio is clicked, a window for displaying the experimental/CFD FTF and fitting will appear.

  1. Data import panel

  Users can set the velocity ratio and then import experimental/CFD FTF from an external Mat file. (It is better to save the data in a Mat file prior to running OSCILOS. The data format is for example shown in the right bottom figure).

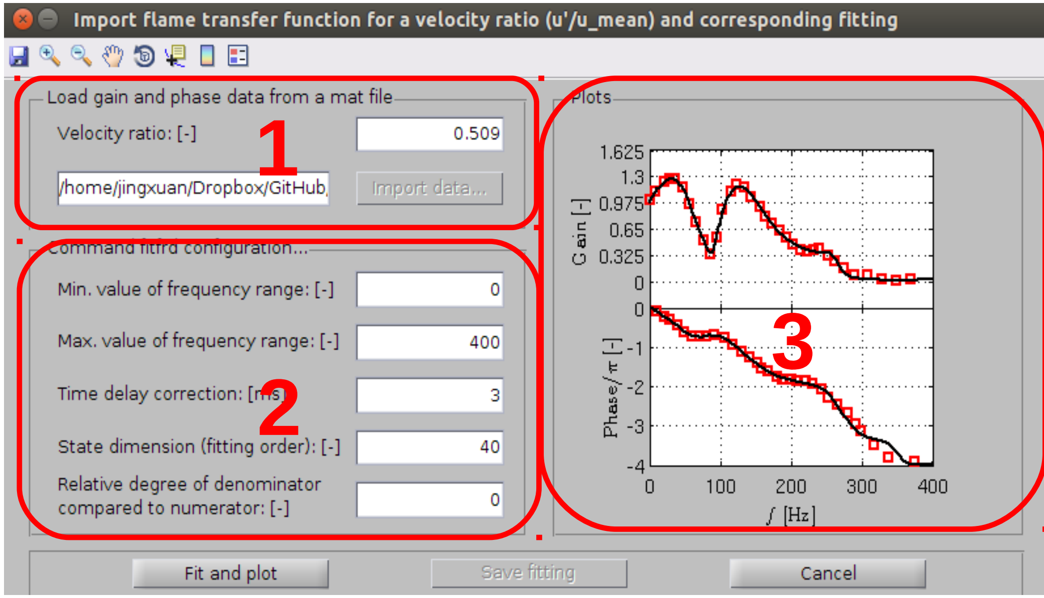
  2. Fitting parameters configuration panel

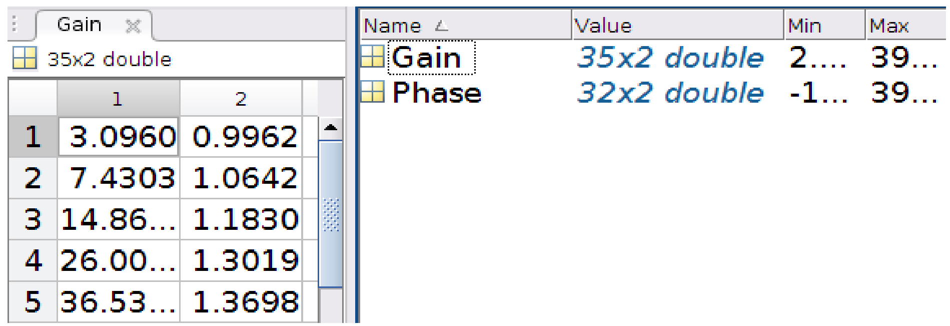
  Users need to set the fitting frequency range, time delay correction, fitting order and relative degree of denominator compared to numerator, which are used for fitting, via the Matlab command "fitfrd".

  3. Plot panel

  The original data (markers) and fitted FTF (solid line) can be viewed from this plot.

- The fitting process is operated upon clicking Fit and plot.
- The original FTF data and fitting parameters are saved by clicking save fitting.





The first column is the frequency and the second one is the corresponding gain or phase lag (in rad).

Jingxuan LI
Department of Aeronautics

Initialization/Boundary conditions

Imperial College London

This menu is used to set the inlet and outlet boundary conditions.

Six kinds of boundary conditions are provided:

✓Open end (R = -1).

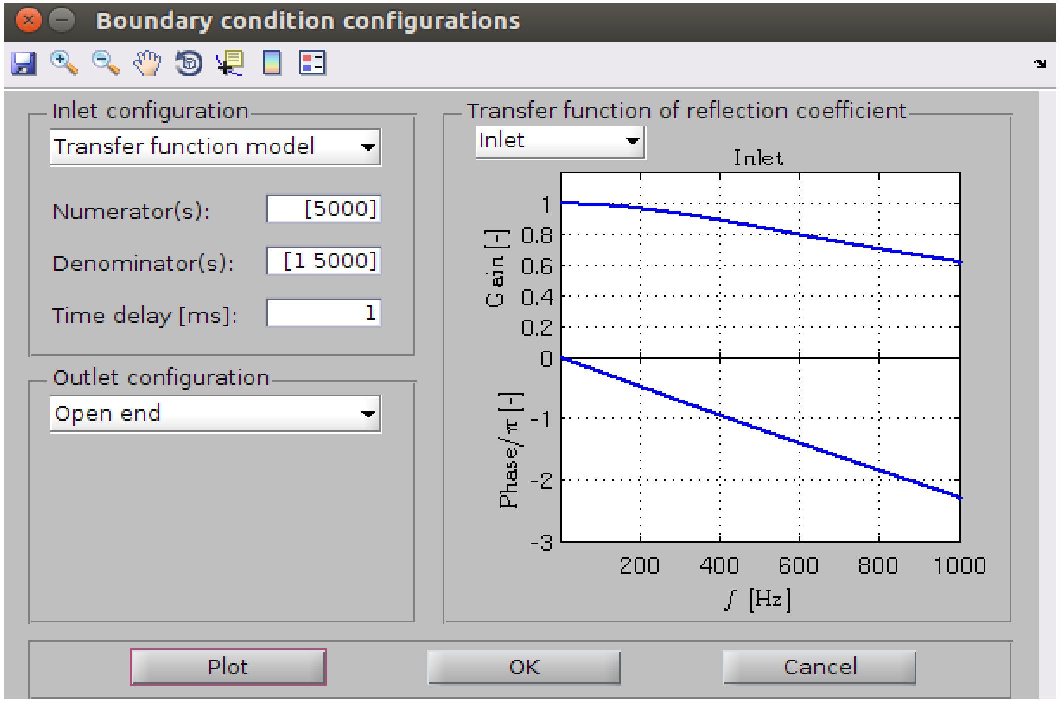
✓Closed end (R = 1).

✓Choked end.

✓User defined (Amplitude and time delay)...

✓User defined (Amplitude and phase)...

✓User-defined model using a polynomial transfer function by inputting the numerator coefficients b and denominator coefficients a. The order of the numerator should not be larger than that of denominator n ≤ m.



$$R(s) = \frac{b_1 s^{n-1} + b_2 s^{n-2} + \cdots + b_{n-1} s + b_n}{a_1 s^{m-1} + a_2 s^{m-2} + \cdots + a_{m-1} s + a_m}$$

Boundary conditions / Include indirect noise from entropy waves —1

Imperial College London

When the user choose the choked outlet boundary, there is an option to include the indirect noise from the entropy or not. Two models accounting for the advection of entropy waves are prescribed in this work.

- A "Rectangular" model (proposed by Sattelmayer (2003)): The p.d.f. or impulse response is modelled as a rectangular pulse of length $2\Delta\tau_C^s$ and height $1/2\Delta\tau_C^s$ centred about the mean residence time τ_C^s:

$$E_C^{\mathrm{inlet}}(t) = \delta(t)$$

$$E_C^{\mathrm{outlet}}(t) = \begin{cases} \dfrac{1}{2\Delta\tau_C^s} & \text{for } \tau_C^s - \Delta\tau_C^s \leqslant t \leqslant \tau_C^s + \Delta\tau_C^s \\ 0 & \text{else} \end{cases}$$

The corresponding Laplace transform of the transfer function between the entropy waves at the outlet and inlet can be expressed as:

$$\frac{\widetilde{E}_C^{\mathrm{outlet}}(s)}{\widetilde{E}_C^{\mathrm{inlet}}(s)} = \tilde{\mathcal{E}}(s)\exp\left(-\tau_C^s\ s\right) = \frac{\exp\left(\Delta\tau_C^s\ s\right) - \exp\left(-\Delta\tau_C^s\ s\right)}{2\Delta\tau_C^s\ s}\exp\left(-\tau_C^s\ s\right)$$

Boundary conditions / Include indirect noise from entropy waves –2

- A "Gaussian" model (proposed by Morgans et al.(2013)): Shear dispersion is assumed to be predominantly caused by spatial variations in the time-mean velocity profile, rather than by turbulent eddies. The impulse response is modelled as a Gaussian distribution:

$$E_C^{\text{inlet}}(t) = \delta(t)$$

$$E_C^{\text{outlet}}(t) = \frac{1}{\sqrt{\pi}\Delta\tau_C^s} \exp\left(- \left(\frac{t - \tau_C^s}{\Delta\tau_C^s} \right)^2 \right)$$

with the Laplace transforms of the transfer function:

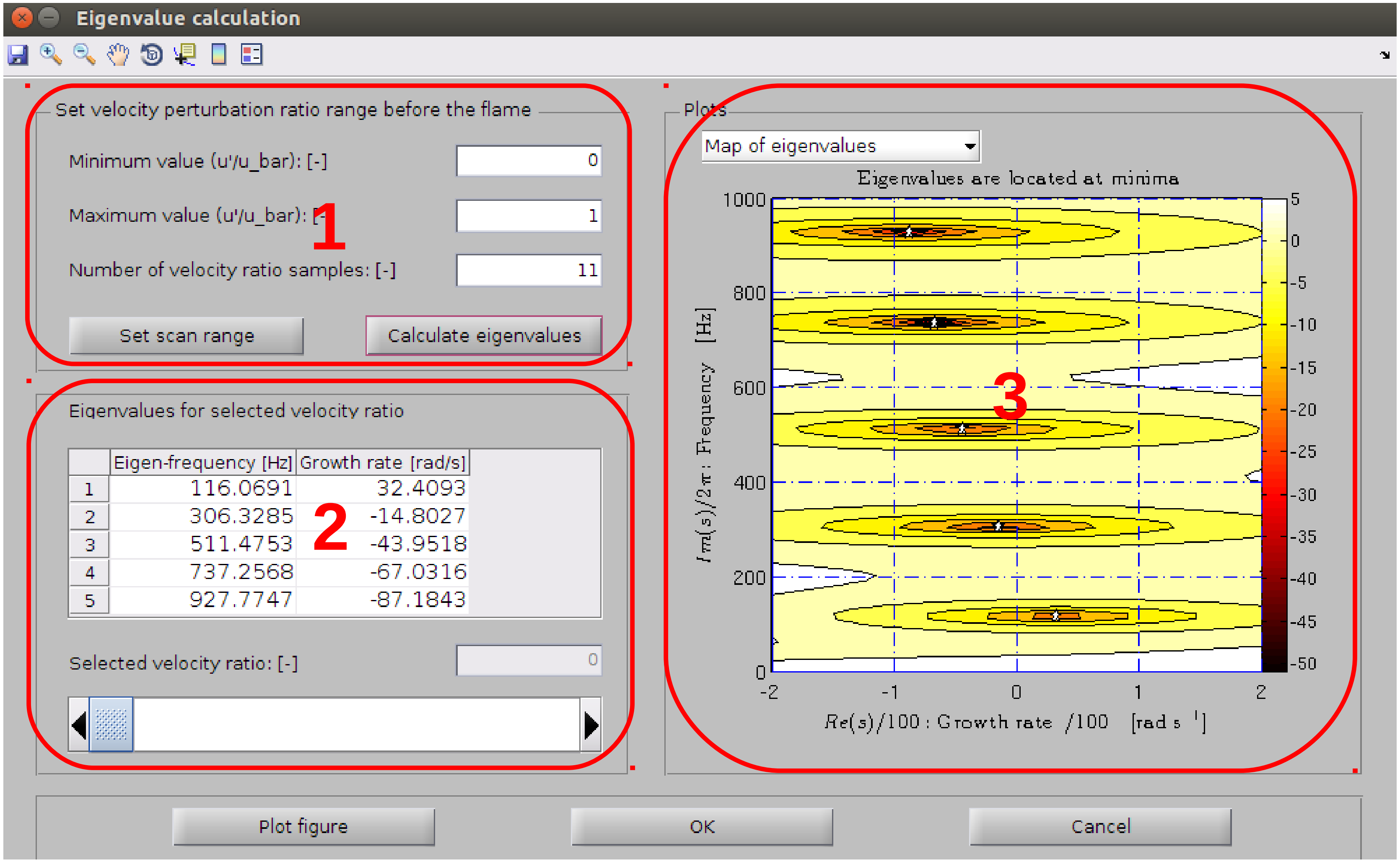
$$\tilde{\mathcal{E}}(s) = \exp\left(\frac{\left(\Delta\tau_C^s \ s\right)^2}{4} \right)$$

where time delay $\Delta\tau_C^s$ is proposed to describe the dispersion of residence time.

Freq. domain analysis / Eigenmode calculation

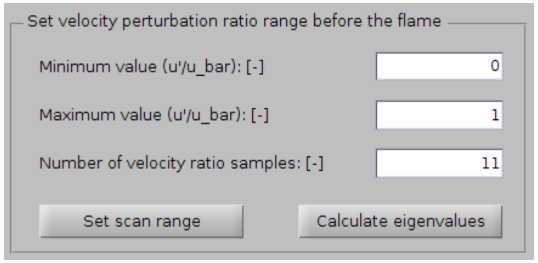
Imperial College London

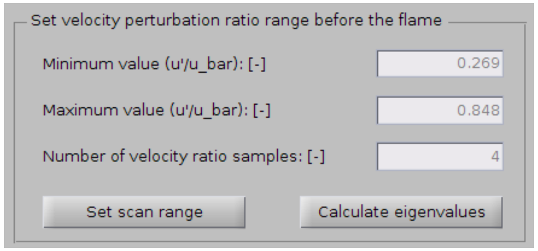
Once all initializations are complete, the user can then progress to the "Eigenmode calculation" panel.

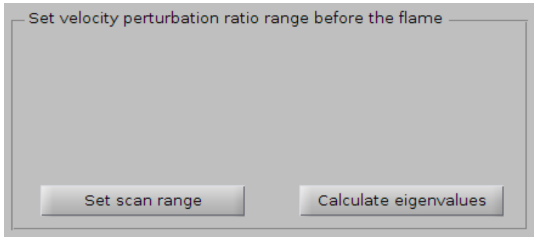


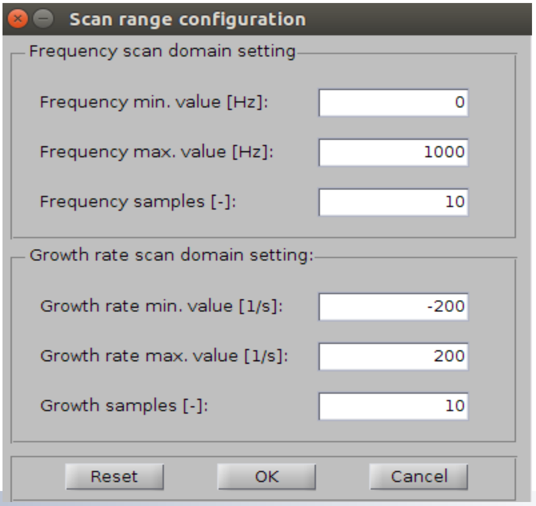
Freq. domain analysis / Eigenmode calculation

- **1. Velocity ratios and eigenmode scan domain configuration panel**
- Users need to set the minimum and maximum velocity ratios before the flame if the nonlinear flame describing function model has been chosen (as shown in first figure). The number of velocity ratio samples is also needed to equally space the velocity ratio range.
- In the case that the flame describing function was provided by experimental or CFD data, these “edit” boxes are not enabled -- their values are automatically assigned (as shown in second figure).
- In the case that linear flame transfer function model was chosen or there is no flame or unsteady heat source inside the combustor, the “edit” boxes are not visible (as shown in the third figure).
- Users need to define a scan range (including frequency and growth rate) to search for eigenvalues within this range, as shown in the right bottom figure. This can be done by clicking “Set scan range”.
- When all the setting has been completed, users can calculate the eigenvalues for a set of velocity ratios (or an arbitrary velocity ratio for linear FTF or no flame case).







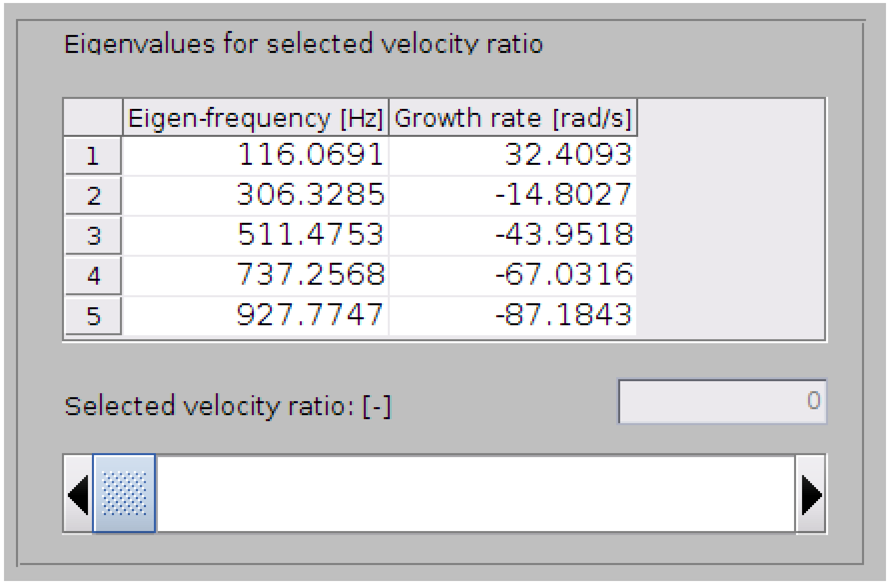


Freq. domain analysis / Eigenmode calculation

Imperial College London

2. Eigenvalues table

- The eigenvalues for a selected velocity ratio (or an arbitrary velocity ratio for linear FTF cases) appears in the table (as shown in the following figure).
- For nonlinear FDF situations, users can change the slider to switch the velocity ratio. The values in the table will automatically change with the velocity ratio.



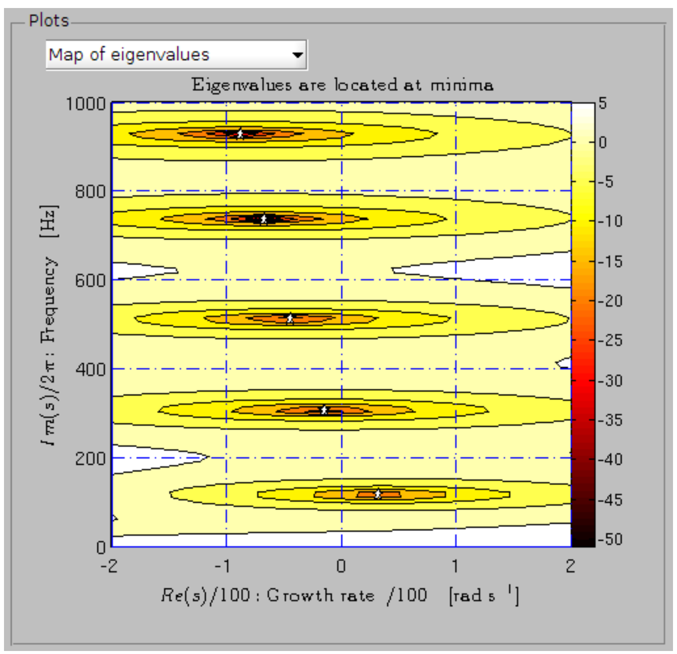
	Eigen-frequency [Hz]	Growth rate [rad/s]
1	116.0691	32.4093
2	306.3285	-14.8027
3	511.4753	-43.9518
4	737.2568	-67.0316
5	927.7747	-87.1843

Imperial College London

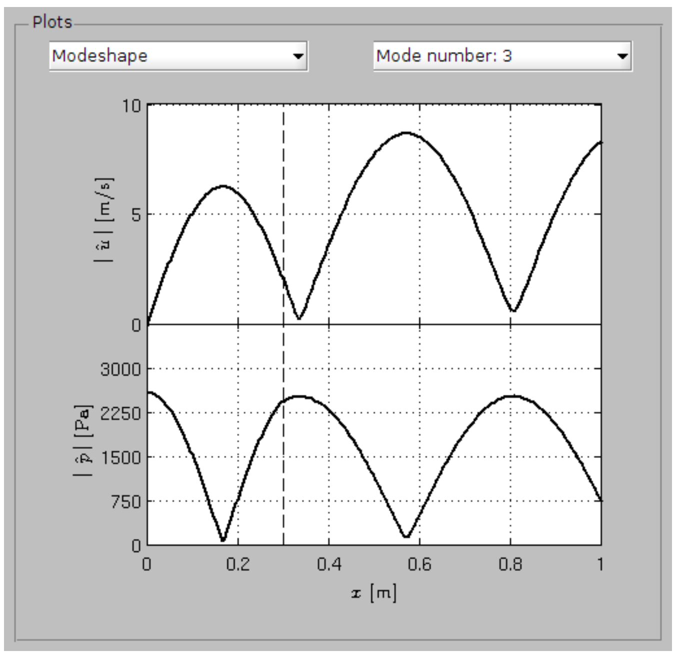
3. Plot panel

✓Users can plot three kinds of figures from the choices of the pop-up menu:

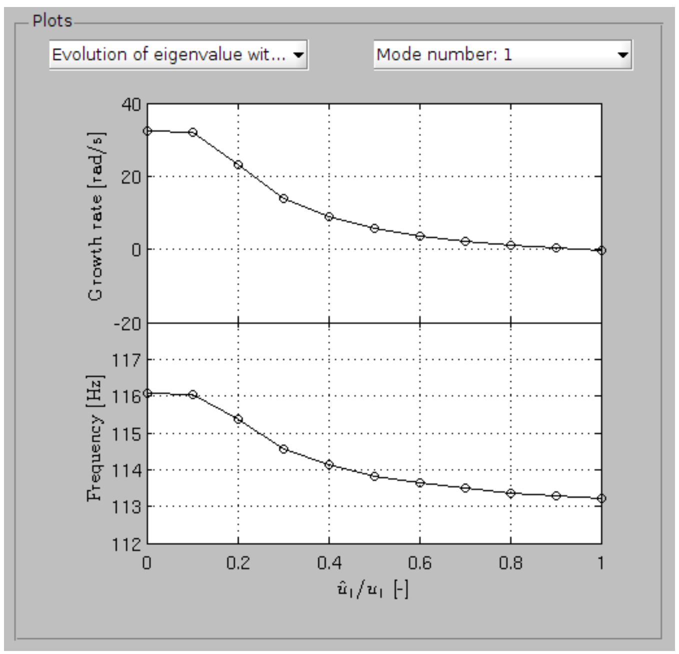
1. A contour map showing the eigenvalue locations (growth rate and frequency).
2. The mode shape (velocity perturbation amplitude (top figure) and pressure disturbance amplitude (bottom figure) distributions along the axial position).
3. The evolution of growth rate (top figure) and eigen-frequency (bottom figure) with increasing velocity ratio for a selected mode. This plot is not available for linear FTF cases.



(a) contour map



(b) mode shape



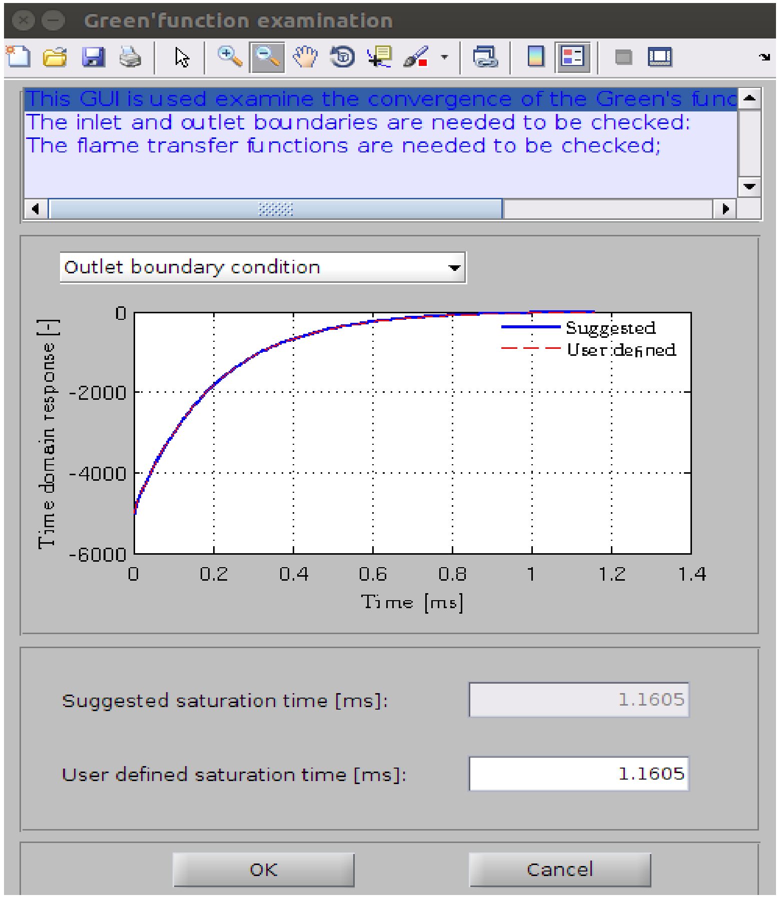
(c) evolution of eigenvalues

Jingxuan LI
Department of Aeronautics

Time domain simulation / Examination of the Green's function

Imperial College London

- When transfer functions have been prescribed, they are convected into Green's function for time domain simulations, by taking the inverse Laplace transform or inverse Fourier transform for causal systems.
- Herein, we only account for the inverse Laplace transform. To avoid calculation errors and reduce the computing time, the absolute value of the inverse Laplace transform of a transfer function should decrease with time and converge relatively quickly.
- This menu option is used to examine the Green's functions of the transfer functions.
- The inlet and outlet boundary conditions may be expressed in transfer function form and need to be checked.
- The flame transfer functions must be examined.
- Users can define the saturation time, to change the computing time and calculation precision.
- The evolution of the Green's function with time can be viewed from the plot.



Jingxuan LI
Department of Aeronautics

Time domain simulation / Parameters configuration

Imperial College London

1. Simulation time and time step

This panel is used to configure the simulation stop time and time step. The minimum time delay is provided and the time step should be smaller than this value.

2. Simulation samples per loop

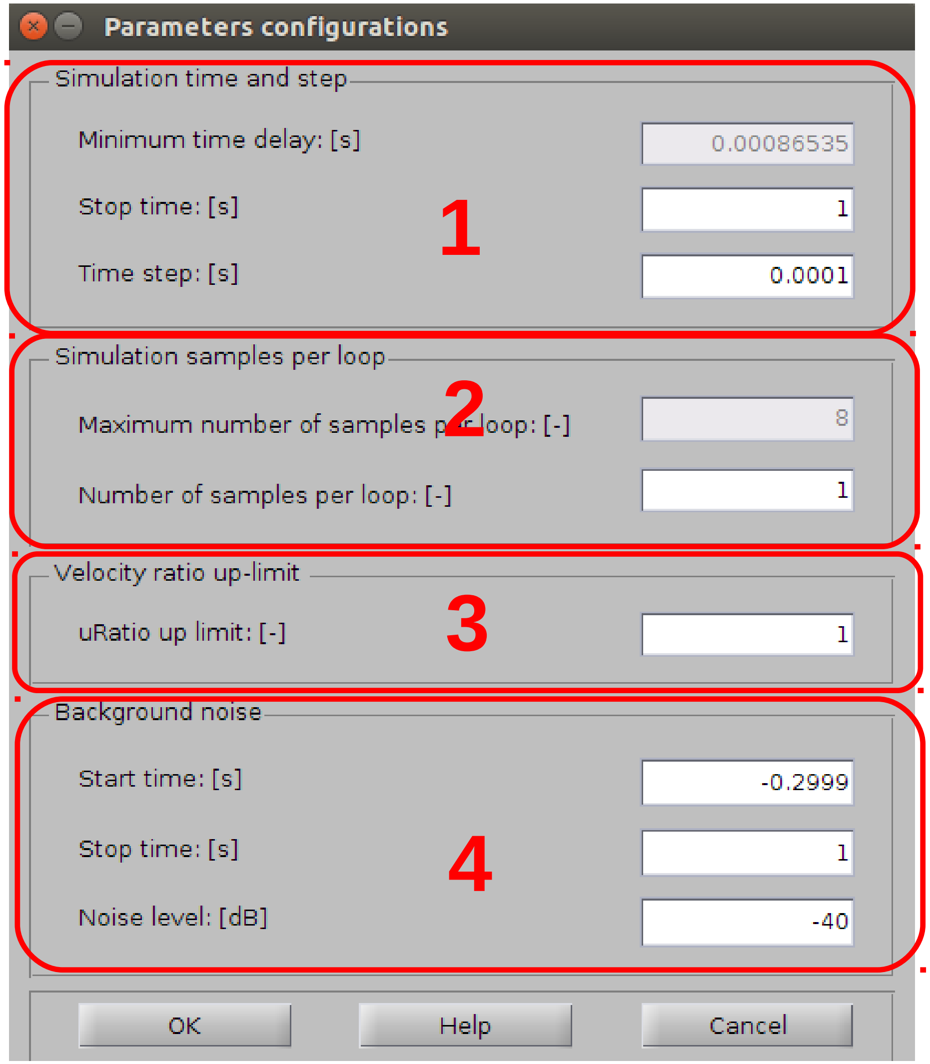
For linear or weakly nonlinear system, the computing speed can be accelerated by increasing the number of time steps in one calculation loop. The maximum number of steps per loop is provided and users can define the number of time steps per loop in the text box. If this box is not active, that means the computing cannot be accelerated and the default value is 1.

3. Velocity ratio upper limit

To avoid signals increasing towards infinity for unstable systems, users need to define an upper limit.

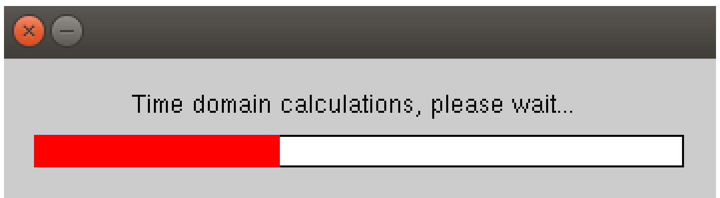
4. Background noise

The background noise information can be defined in this panel. These noises can be used to stimulate any instabilities within the combustor.

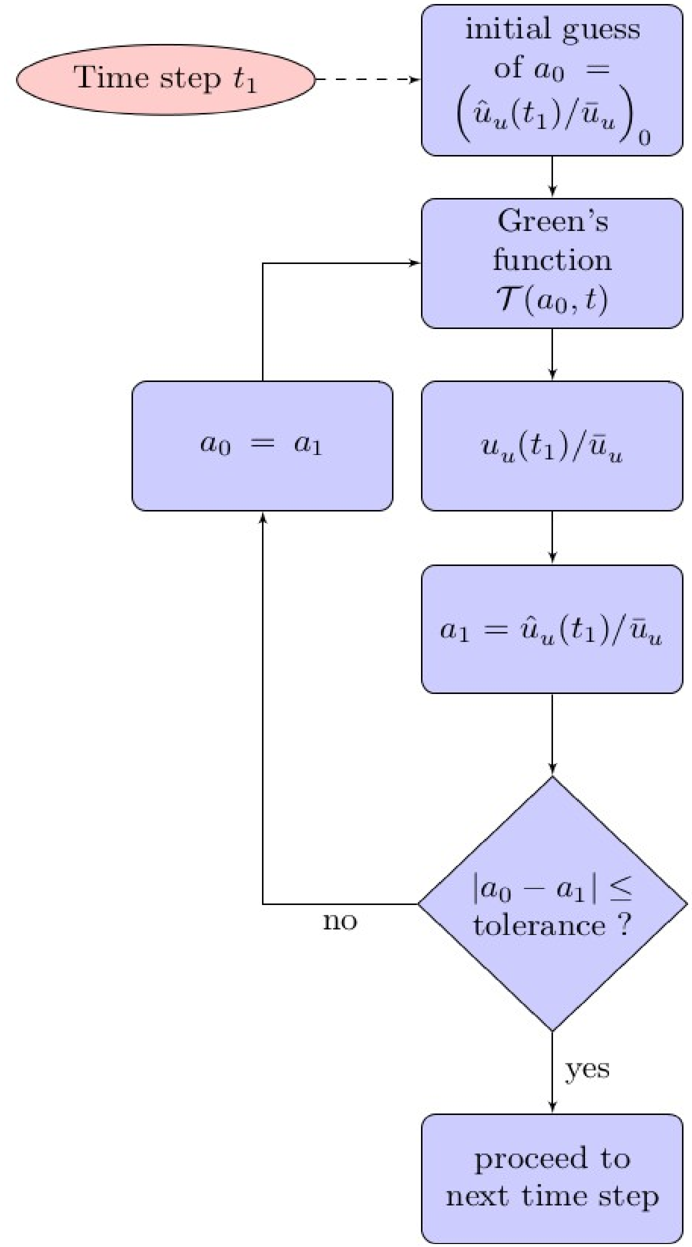


Time domain simulation /Simulation…

- Once all initializations are complete, the user can then progress to the "simulation…" panel.
- For linear systems or nonlinear systems with an abrupt saturation limit (Dowling's model), the calculation is simpler and a wait-bar box appears to show the computing progress.



- The calculation becomes more complicated when users have chosen flame describing function models (such as our model). The transfer function changes with velocity ratio and the Green's function should be updated every time step based on the velocity ratio at the corresponding time step. However, calculating the velocity ratio needs knowledge of the Green's function.
- The calculation method can then be summarized using the flow chart on the right.

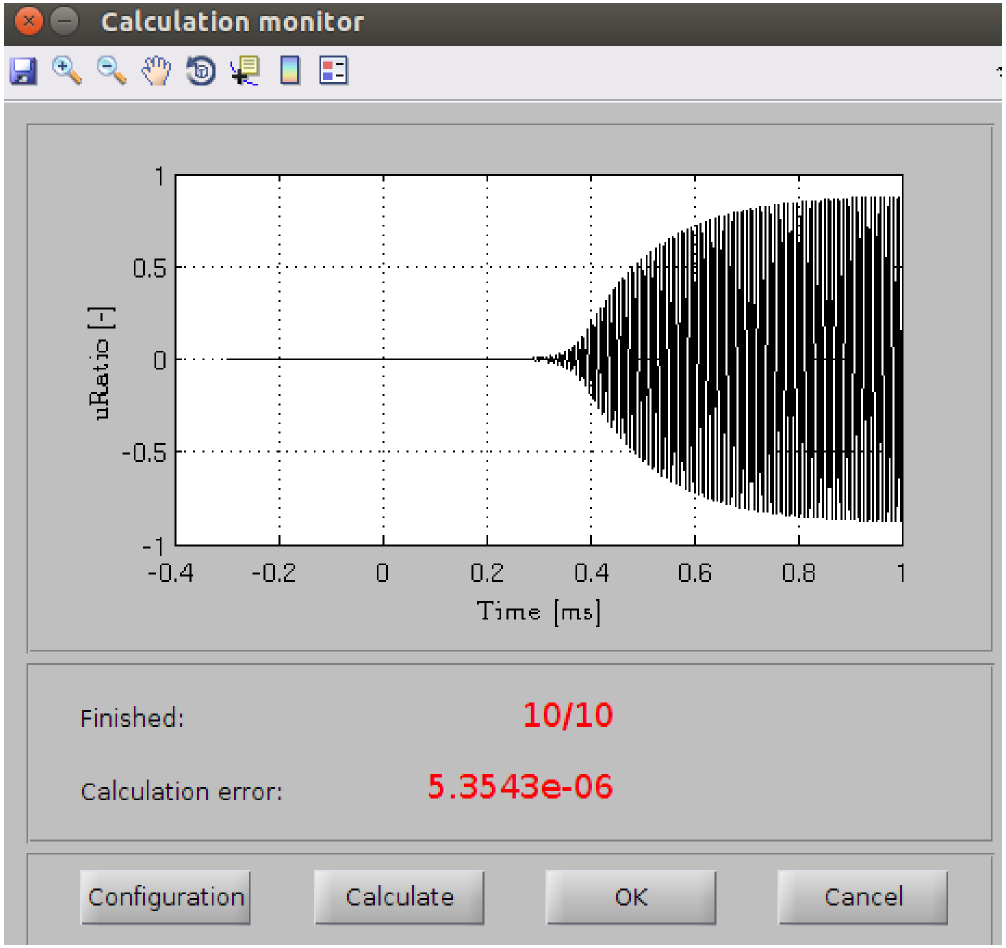


Jingxuan LI
Department of Aeronautics

Time domain simulation /Simulation…

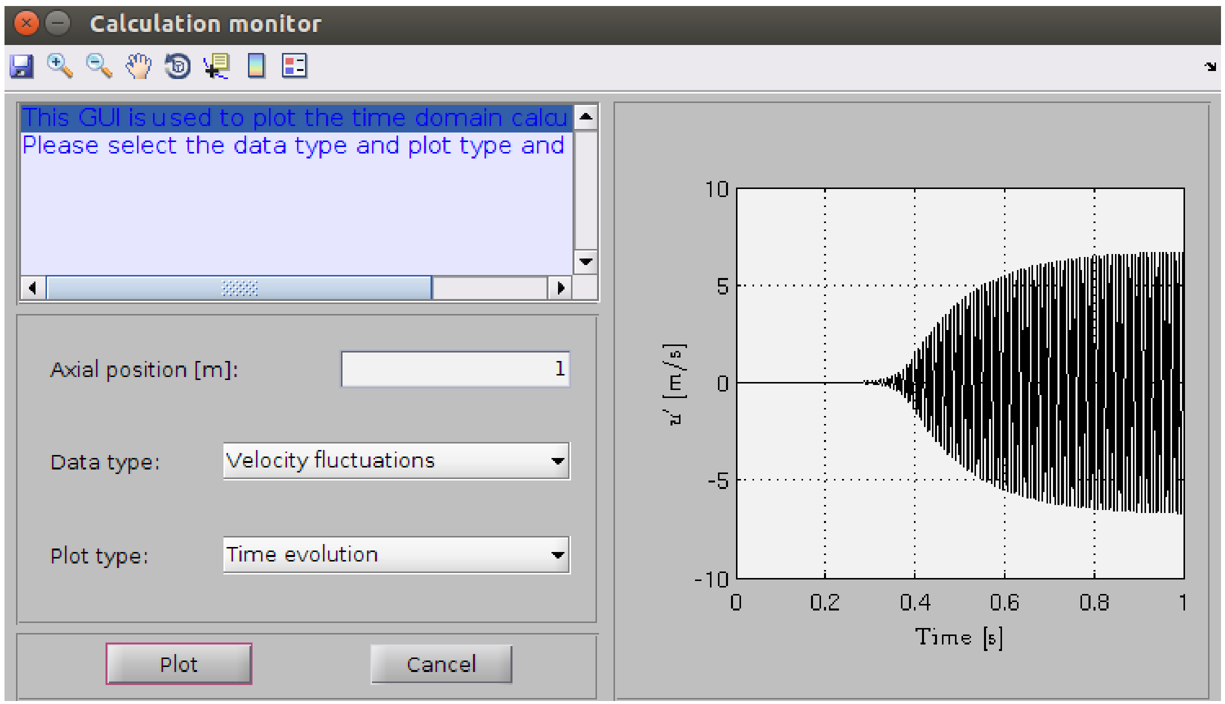
Imperial College London

- When users have chosen a flame describing function model (such as our model), a calculation monitor window, as shown in the right figure, appears to show the calculation progress.
- The evolution of velocity ratio before the flame is plotted in the figure.
- The calculation error is also shown.



Time domain simulation /Results output and plots

- This menu option is used to plot the time domain simulation results.
- The time evolution and power spectrum of the selected signals can be plotted.



Example 1 A cold open tube

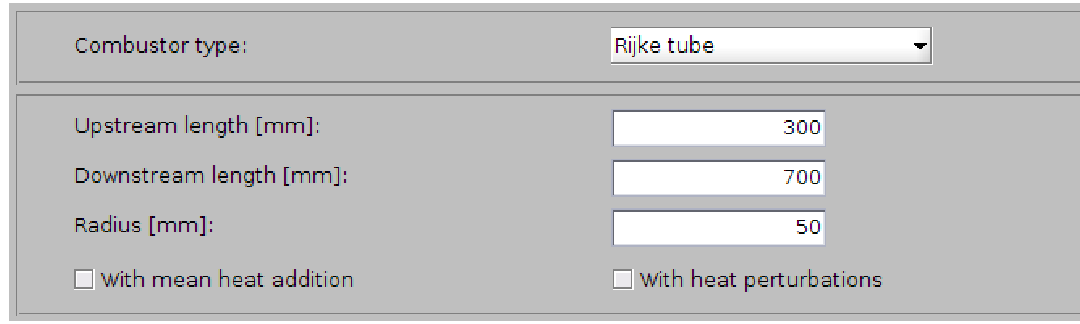
➢ Users can directly load the file "Case_A_cold_open_tube.mat" from the "cases" folder to see the detailed configuration and results.

1. Combustor dimensions

The combustor type is set to Rijke tube.

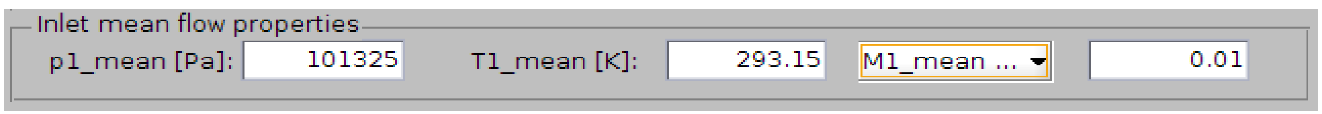
The dimensions are shown in the right figure.

There is no heat addition in the tube.



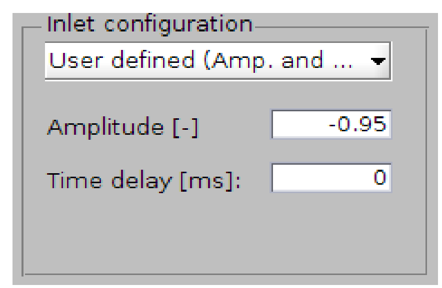
2. Mean flow and thermal properties configuration

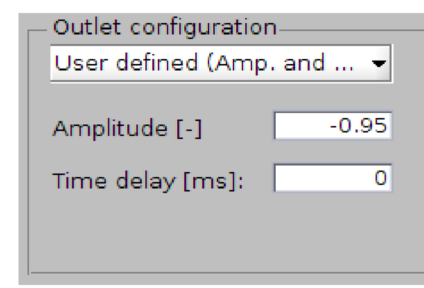
Since there is no heat addition, the panel for heat addition configuration is not visible. The mean flow properties at the inlet are shown in the following figure.



3. Boundary conditions

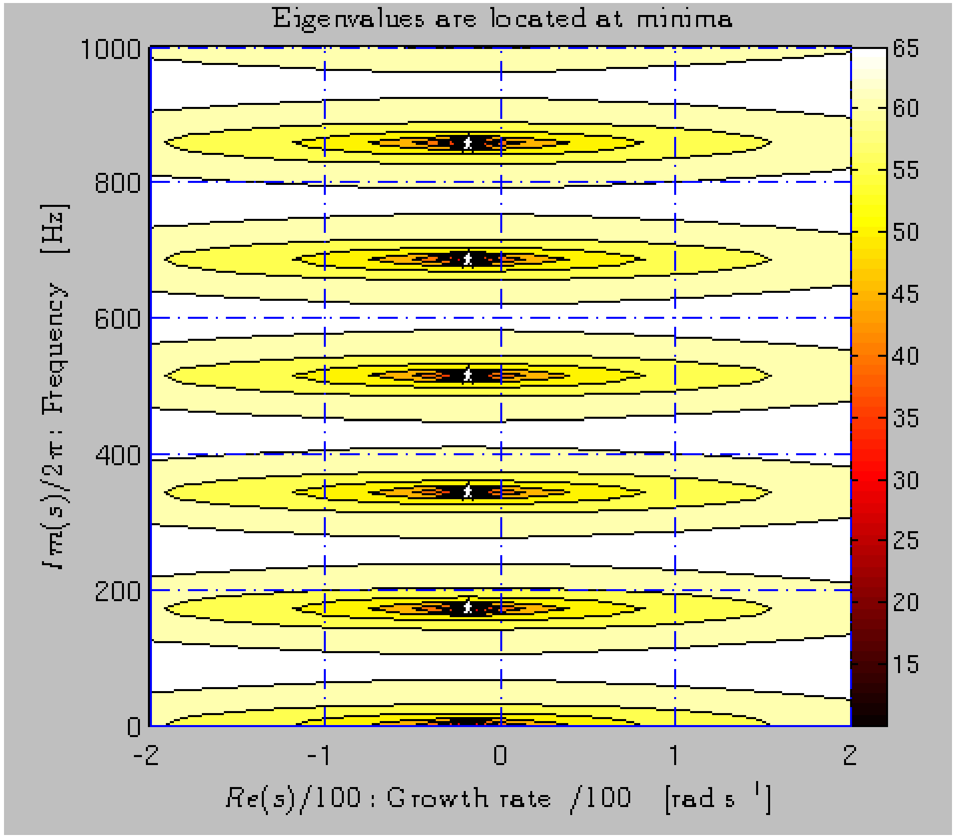
Since there is no heat perturbation, the user directly progresses to the boundary conditions configuration panel. The inlet and outlet are set to open and the pressure reflection coefficients are set to negative constant values, as shown in the right figures.





Example 1 A cold open tube

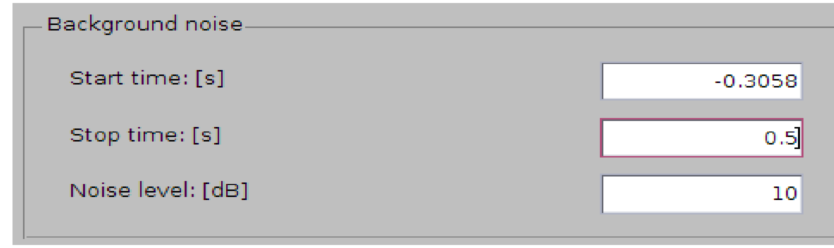
- The eigenvalues and their distributions in the s-plane are calculated during the frequency domain analysis. The contour map showing these eigenvalues is shown in the bottom figure. The first five modes of the system are on the left side of the s-plane indicating that the system is stable.



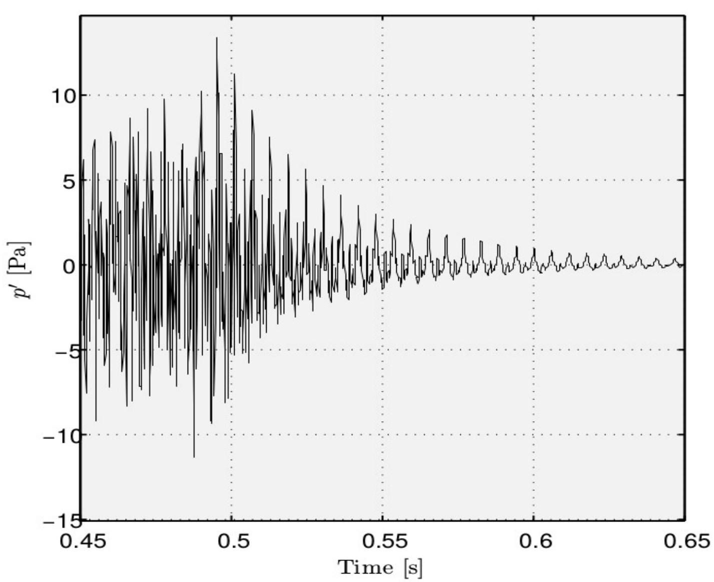
	Eigen-frequency [Hz]	Growth rate [rad/s]
1	171.6059	-17.6045
2	343.2118	-17.6045
3	514.8177	-17.6045
4	686.4237	-17.6045
5	858.0296	-17.6045

Example 1 A cold open tube

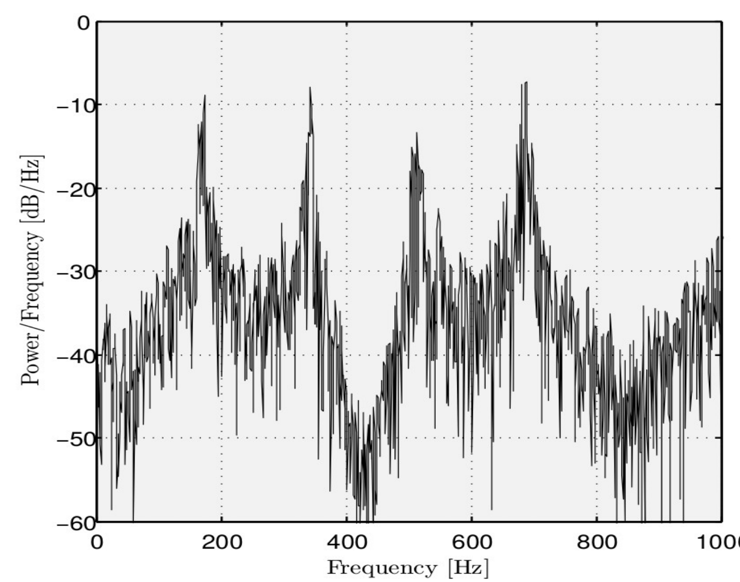
- We now progress to time domain simulations. The background noise configuration is shown in the following figure.
- The white noise has a power of 10 dB and stops at t = 0.5 s.



- The time evolution and power spectrum of the pressure perturbations at the axial position x = 0.6 m are shown in the following figures. In the presence of additional white noise, all of the excited modes are stable and disturbances are attenuated. As shown in the PSD figure, the frequencies of the peaks are the same as those predicted with the frequency domain analysis.



(a) time evolution



(b) PSD

Example 2 Hot combustor

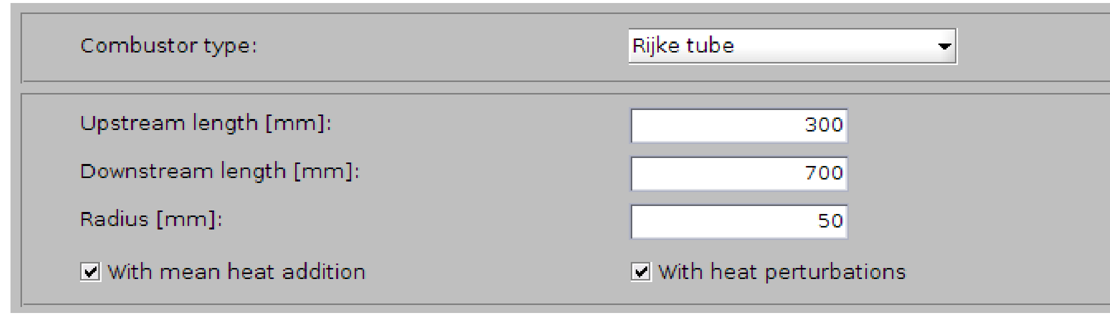
Imperial College London

➢ Users can directly load the file "Case_Hot_tube.mat" from the "cases" folder to see the detailed configuration and results.

1. Combustor dimensions

The combustor type is set to Rijke tube.

The dimensions are shown in the right figure.

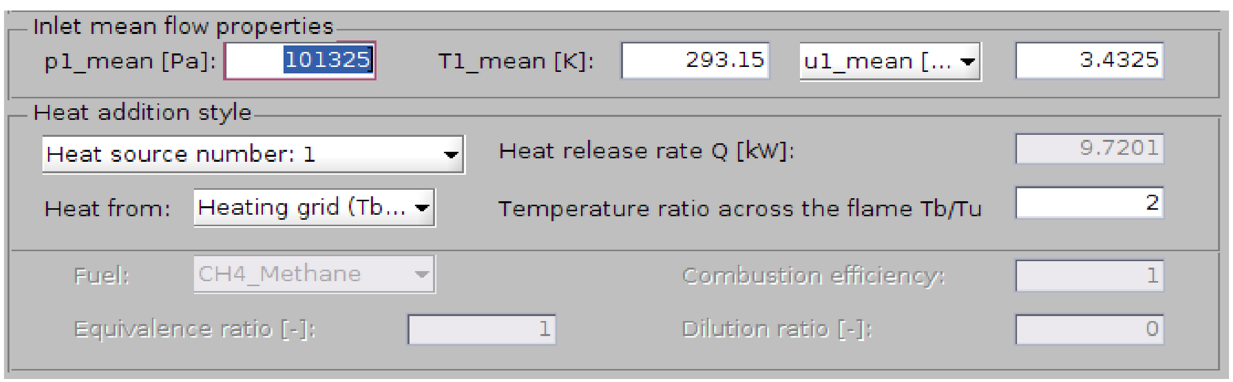


2. Mean flow and thermal properties configuration

The mean flow properties at the inlet are shown in the following figure.

Heat is from a heat grid and the temperature ratio is set to 2.

The mean heat release rate is calculated as 9.7201 kW.

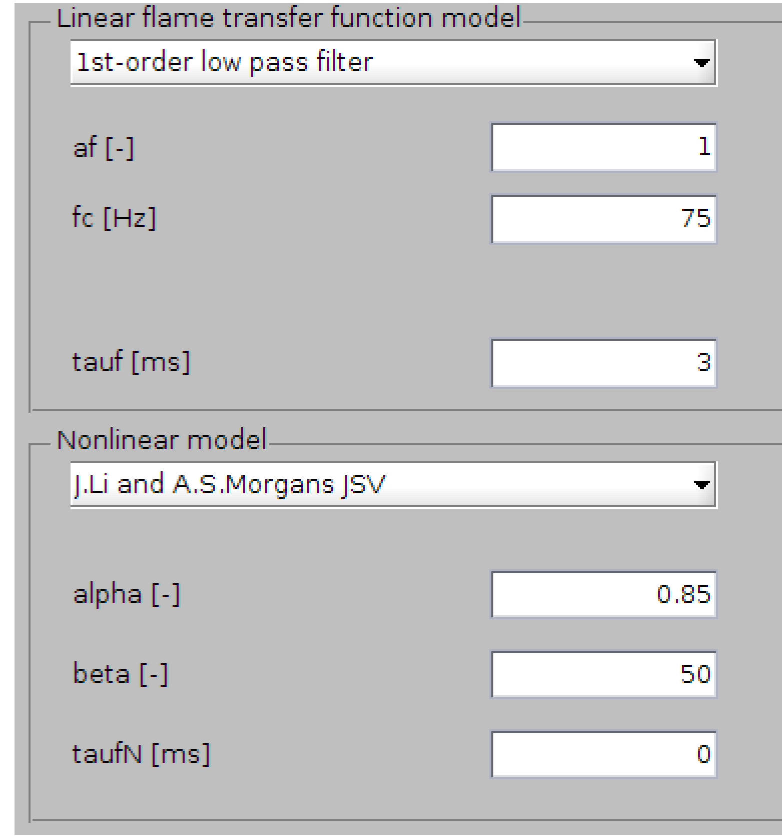


Example 2 Hot combustor

3. Flame model

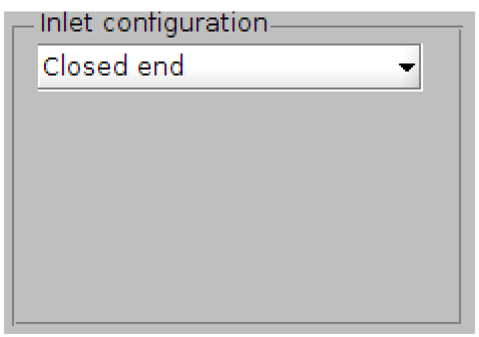
Our nonlinear flame describing function is chosen as the flame model.

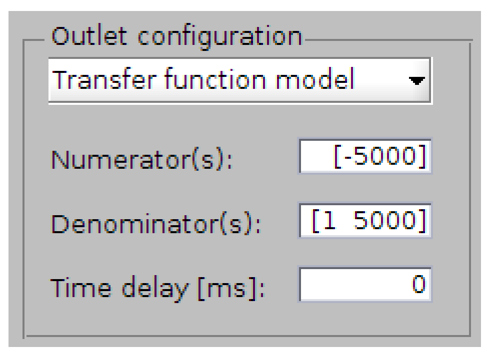
The parameters configuration for the flame transfer function and the nonlinear saturation model are shown in the right figure.



4. Boundary conditions

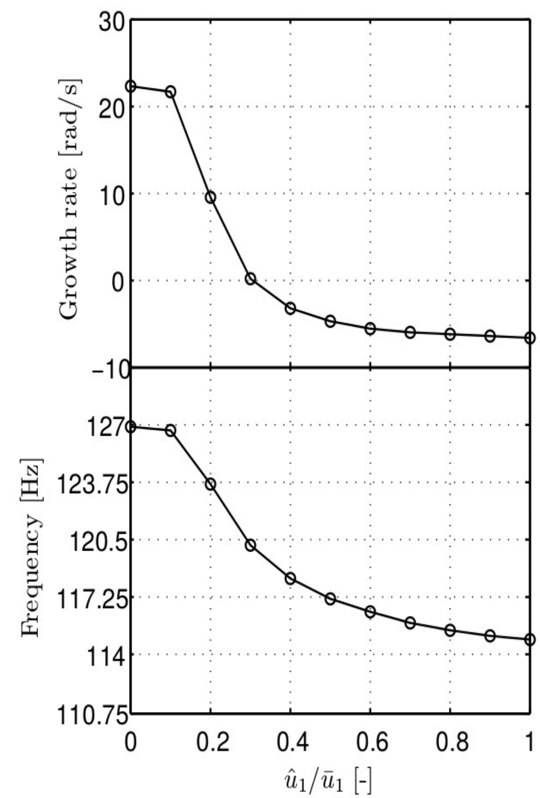
The inlet is set to closed and the outlet is set to open. The reflection coefficient is expressed as a transfer function as shown in the bottom figure.

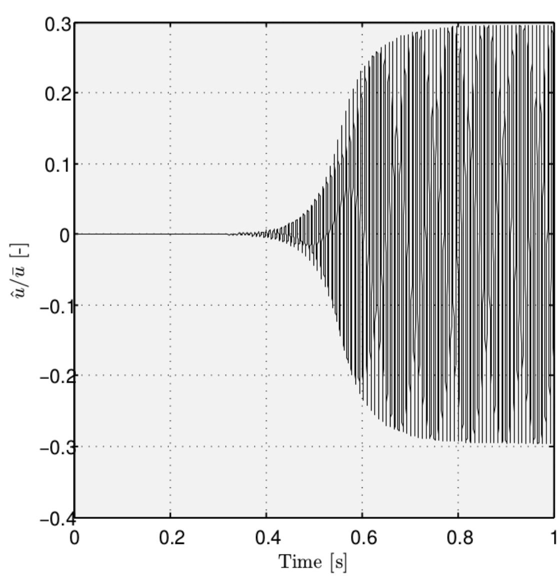


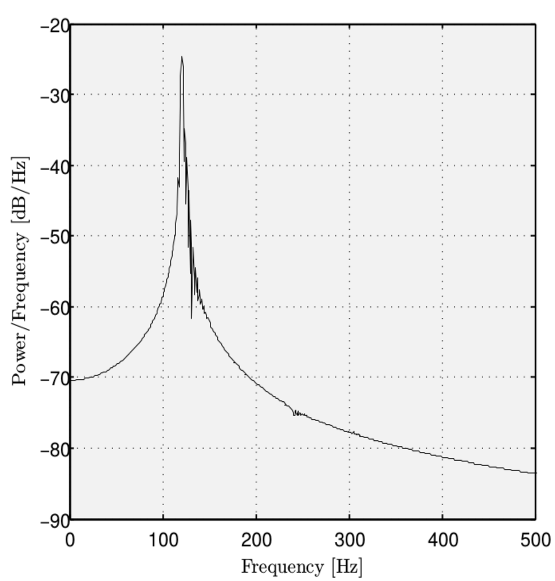


Example 2 Hot combustor

- The right figure shows the evolution of growth rate and resonant frequency of the first mode with velocity ratio before the flame. With increasing normalized velocity perturbations, the growth rate decreases and a limit cycle is finally established when the velocity ratio equals to 0.3.
- This has been validated in the time domain simulation results. The figures at the bottom show the evolution of velocity ratio with time.



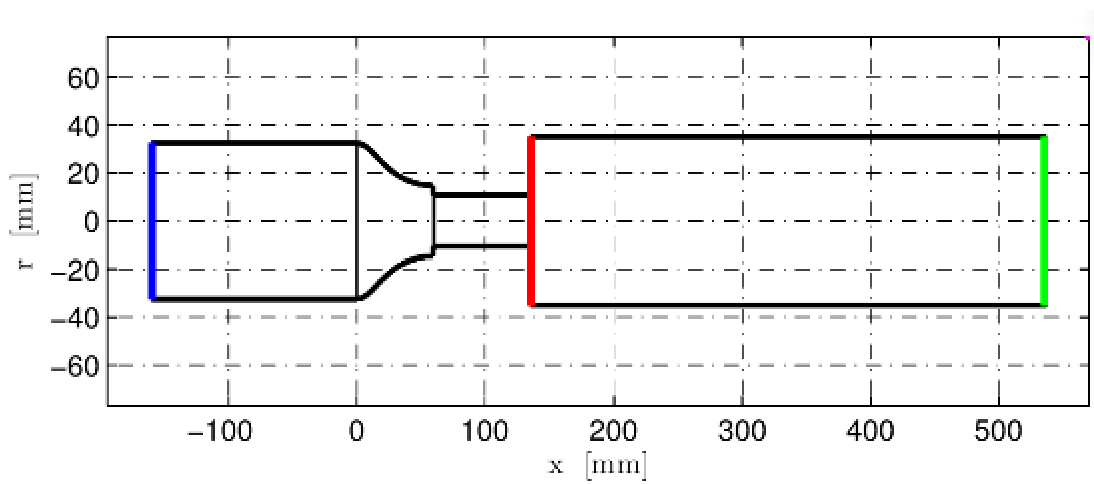


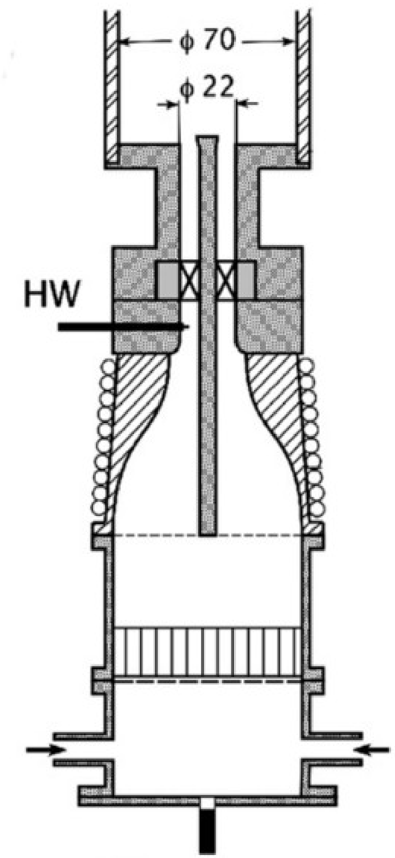


Example 3: A laboratory combustor rig

Imperial College London

- The experiments were carried out by Palies and co-workers in Laboratory EM2C. The combustor includes a plenum, an injection unit and a combustion chamber terminated by an open end. The compact flame is stabilized at the beginning of the combustion chamber.
- OSCILOS can account for this complicated combustor shape and the combustor shape can be viewed from the following figure.
- Experiments were carried out with the plenum and chamber comprising varying lengths to change the eigenvalues of the combustor. Herein, we only take one unstable case for the comparison between the calculation results from OSCILOS and the experimental results.

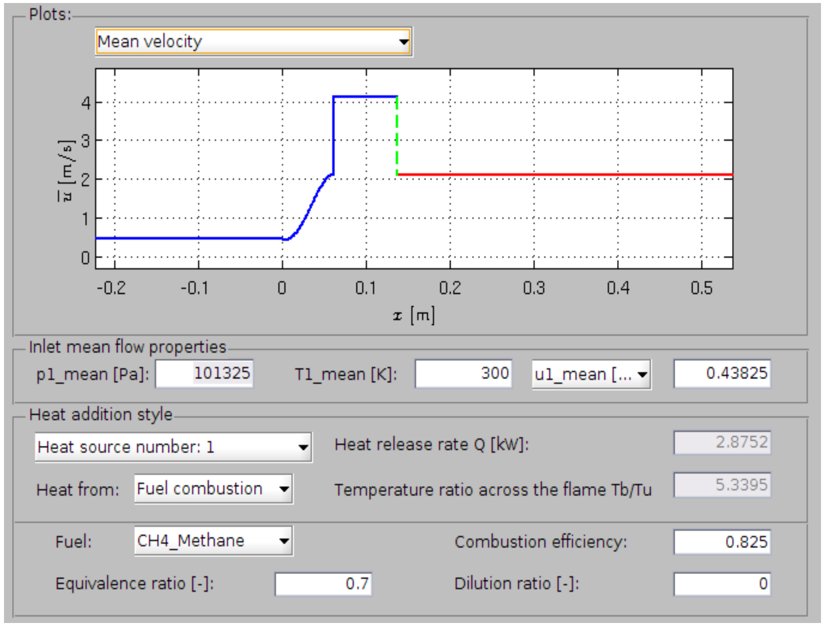


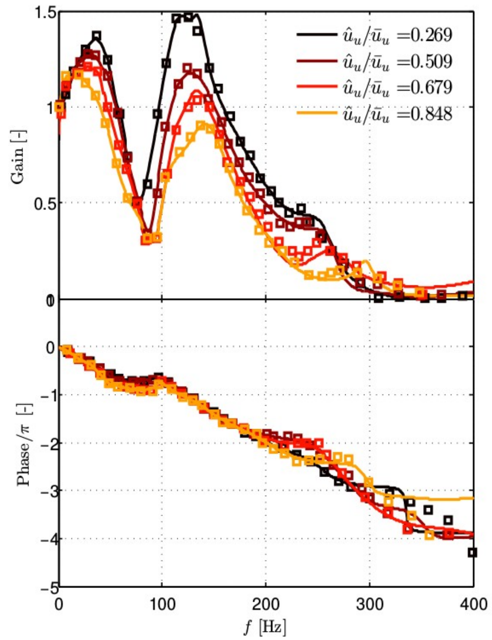


Example 3: A laboratory combustor rig

Imperial College London

- Mean flow and thermal properties: methane is used as the fuel and the equivalence ratio is 0.7. The measured mean temperature of the burned gases is 1600 K. So that the calculated mean temperature matches the experimental result, the combustion efficiency is set equal to 0.825.

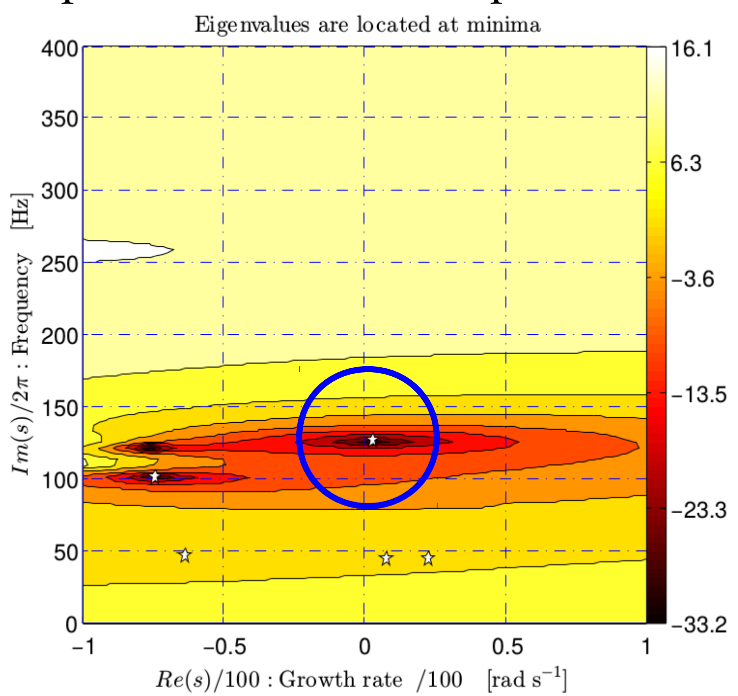




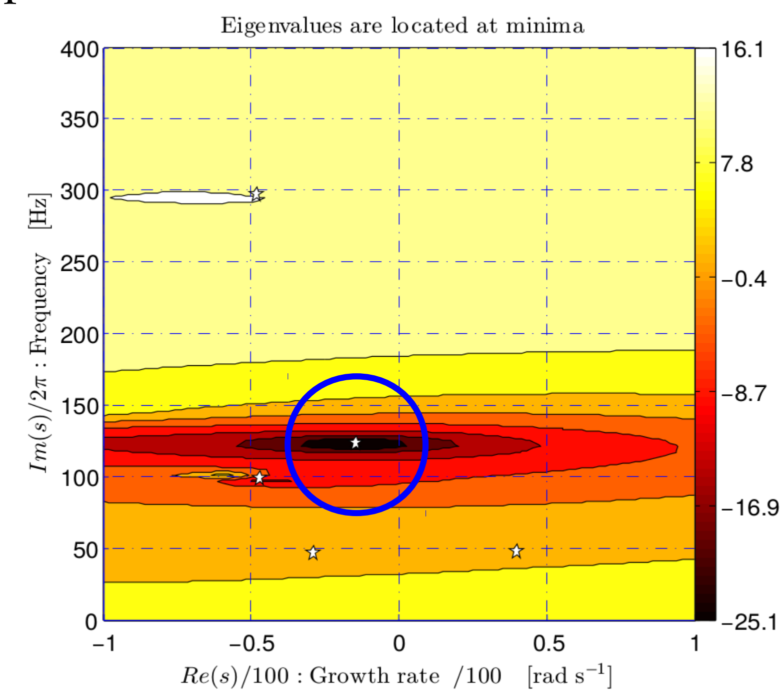
- Flame model: the experimental flame describing functions (markers) and their fitted results (solid lines) are shown in the right figure.
- Boundary conditions: the inlet boundary condition is set to closed with a reflection coefficient of 0.97, and the outlet is set to open.

Example 3: a laboratory combustor rig

- The distribution of eigenvalues are shown in the following contour maps for two velocity ratios. The main unstable modes are highlighted by the blue circles.
- With the increase of flow velocity, the growth rate of the unstable mode decreases to zero and a limit cycle is established. The predicted resonant frequency and velocity ratio before the flame when the limit cycle is established are 126.1 Hz and 0.714, respectively. In the experiment, the resonant frequency and velocity ratio before the flame when the limit cycle is established are 126 Hz and 0.68, respectively.
- The prediction matches quite well the experimental results.



(a) $\hat{u}_u/\bar{u}_u = 0.68$



(b) $\hat{u}_u/\bar{u}_u = 0.85$