Tutorials

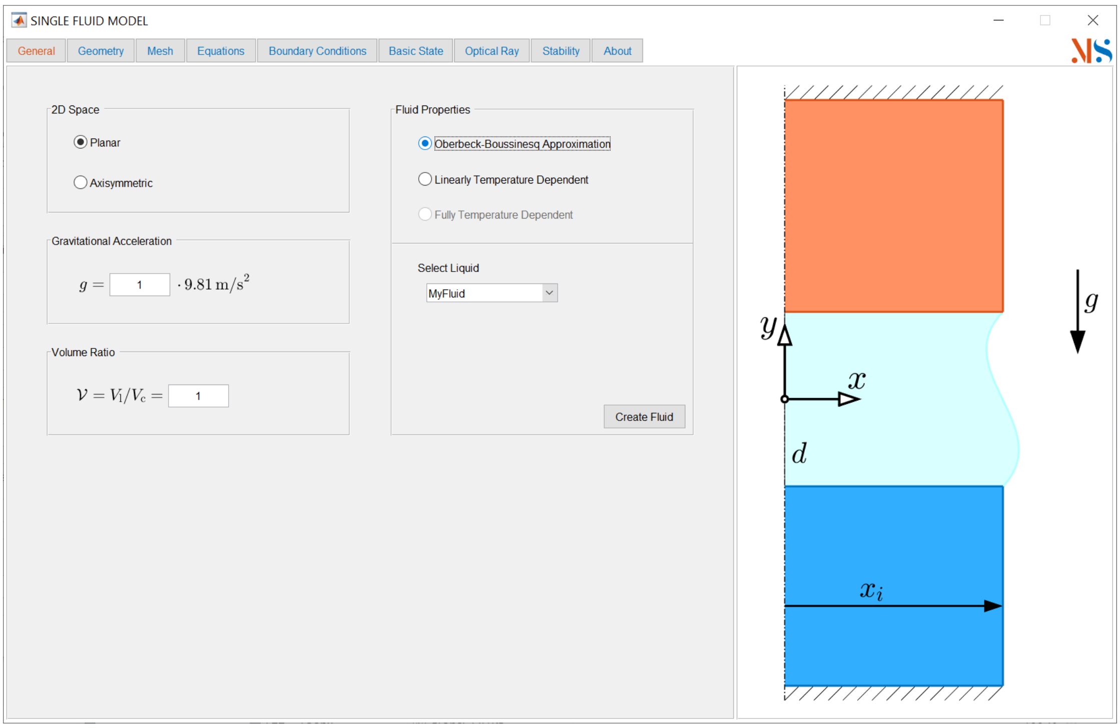
The tutorials presented hereinafter are intended as a guide to the user with the main features implemented in `MaranStable`. The GUI `MaranStable` is started by running the file `main.m` with Matlab 2022 or a more recent version.

1 Rayleigh–Bénard instability

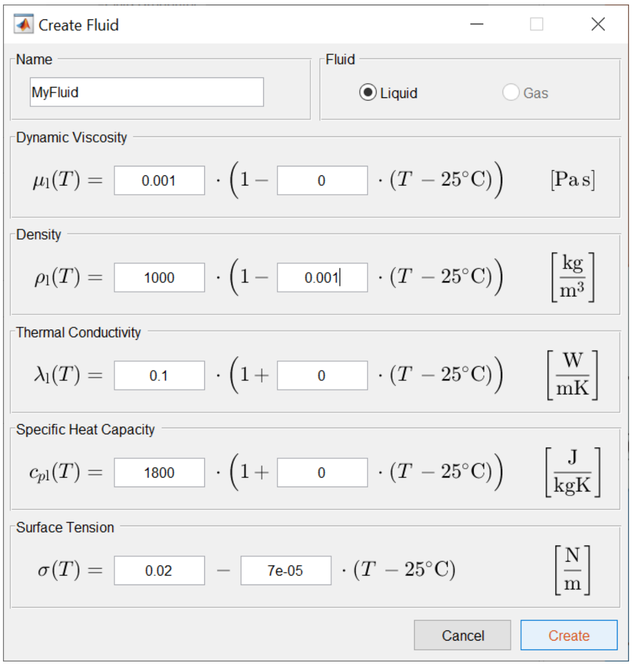
The first tutorial deals with the Rayleigh–Bénard problem, i.e. the instability of a quiescent fluid layer heated from. Here we consider a planar geometry laterally confined by slip walls as a substitute for an infinite layer. This tutorial corresponds to the file `example_1.mat` in the folder `tutorials`.

- Select the single-phase solver

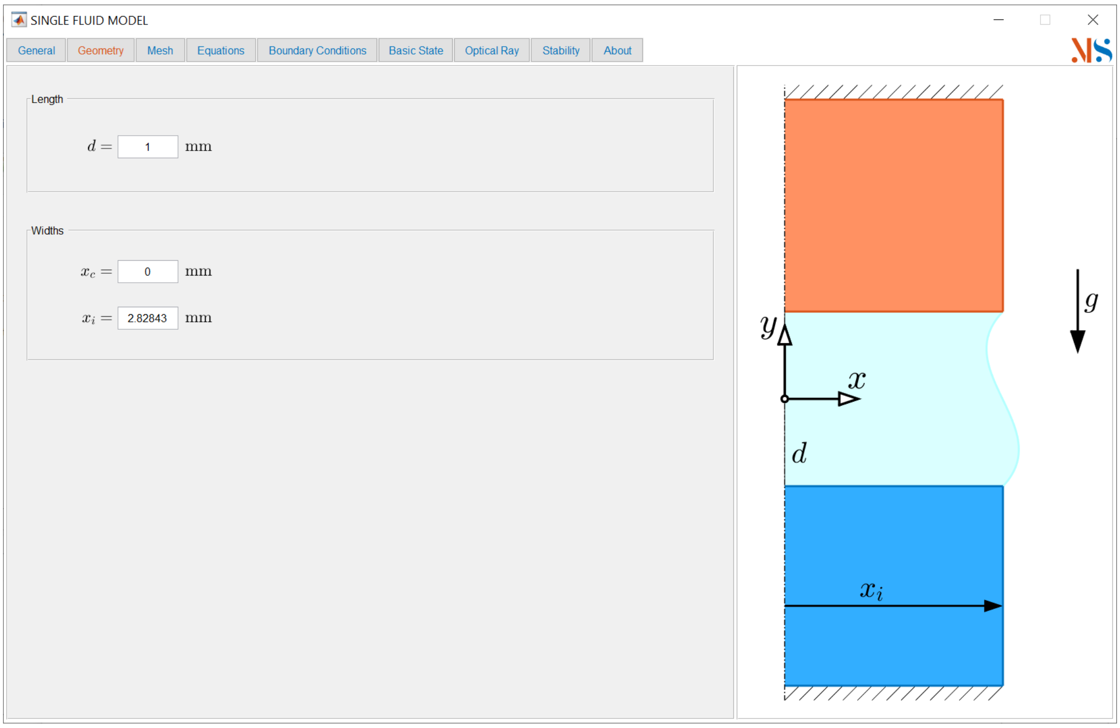
- Set up the planar geometry and the gravitational acceleration. Since we consider a rectangular domain, the volume ratio is set to $\mathcal{V} = 1$. Select the Oberbeck–Boussinesq approximation as physical model.



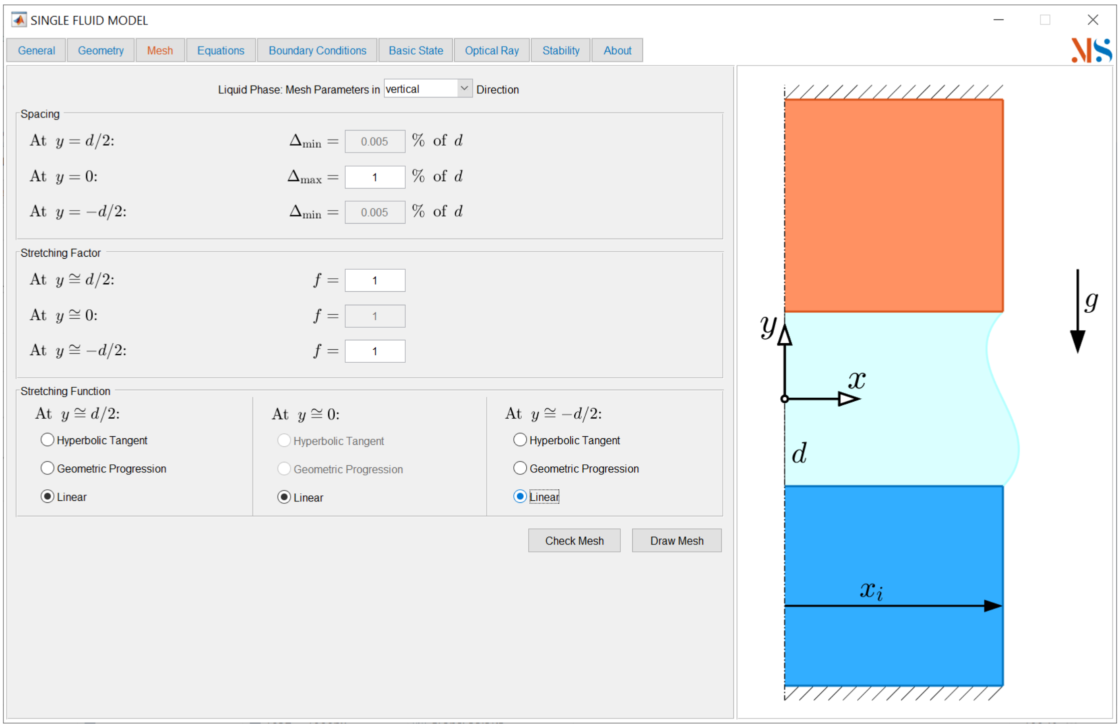
- Set up a new fluid by clicking on `Create Fluid`. The parameters are set as follows. Then click `Create`.

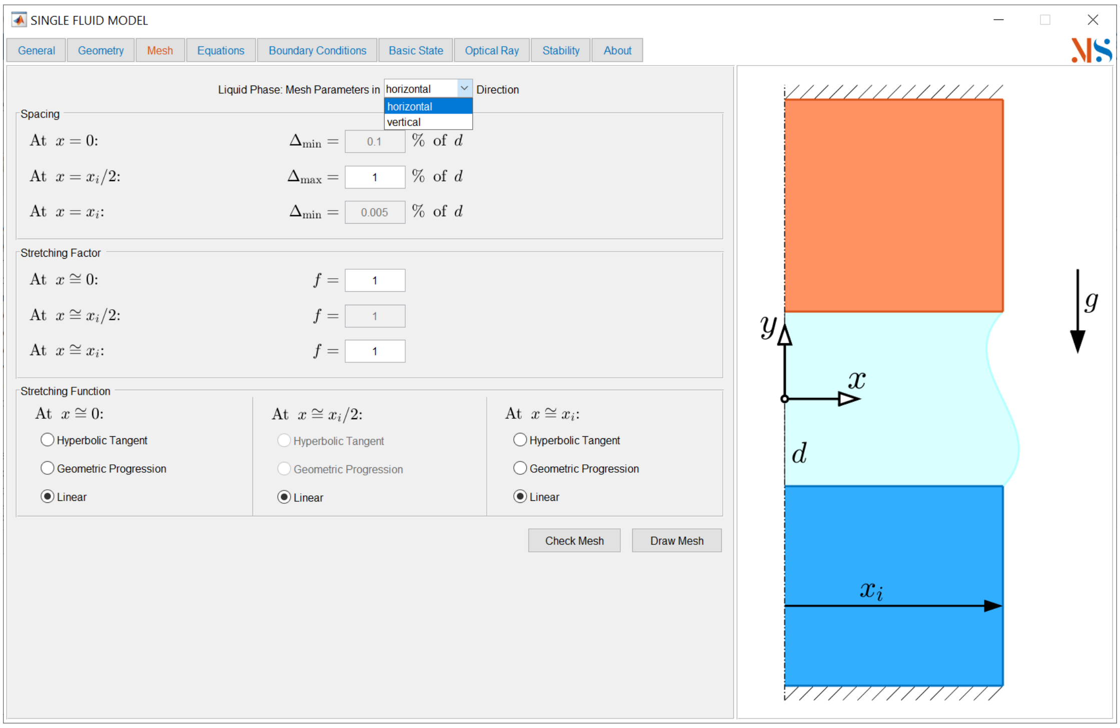


- Set up the planar geometry as follows. The values are selected to coincide with the critical wavelength ($\lambda_c = 2\sqrt{2} = 2.82843$) of the Rayleigh–Bénard instability for free slip on the horizontal boundaries.

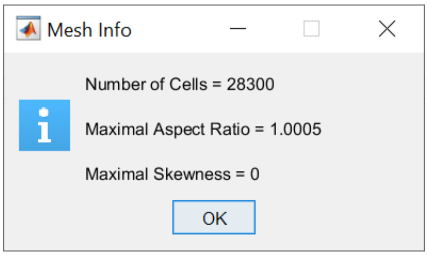


- Set up the mesh as shown below (both horizontal and vertical coordinates must be discretized). You can change the discretization parameters if a finer resolution is needed.

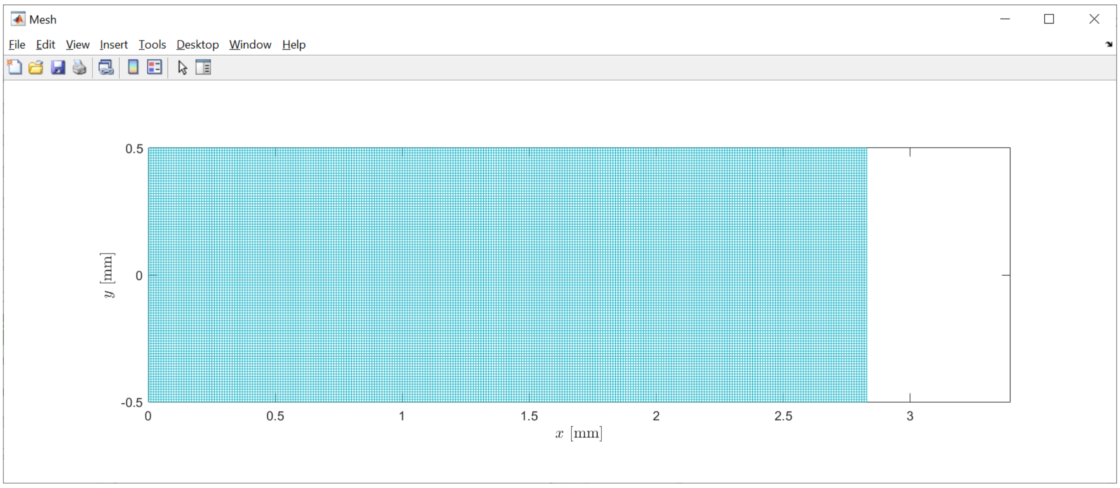




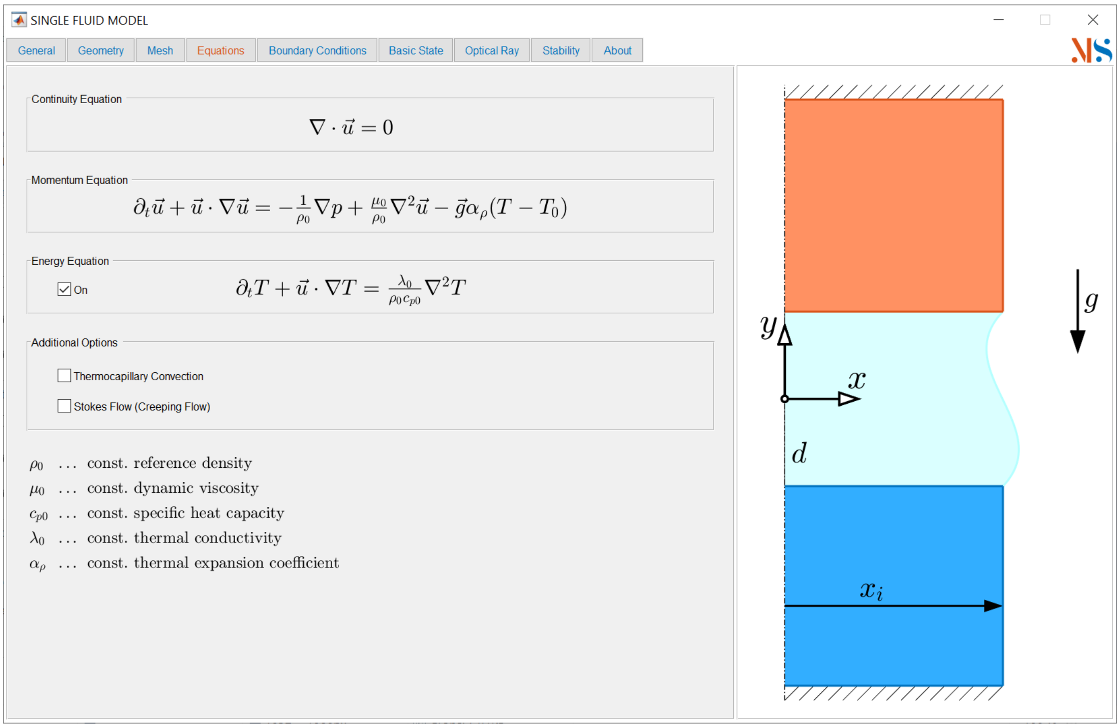
- Click on `Check Mesh`



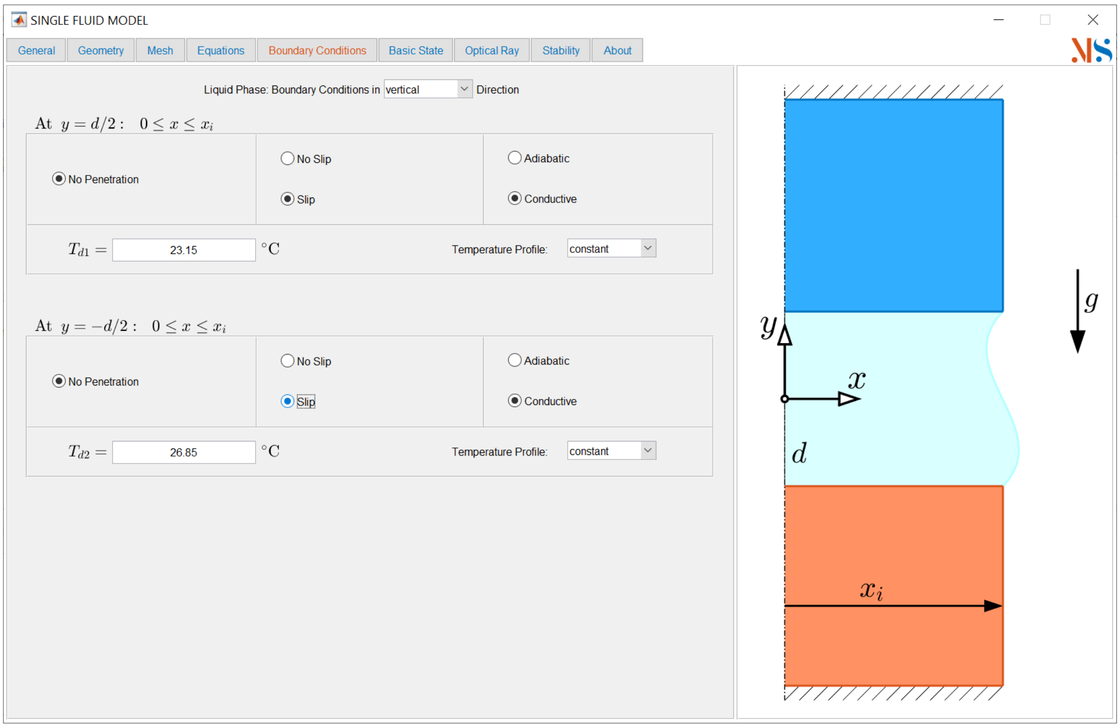
- Visualize the mesh by clicking `Draw Mesh`

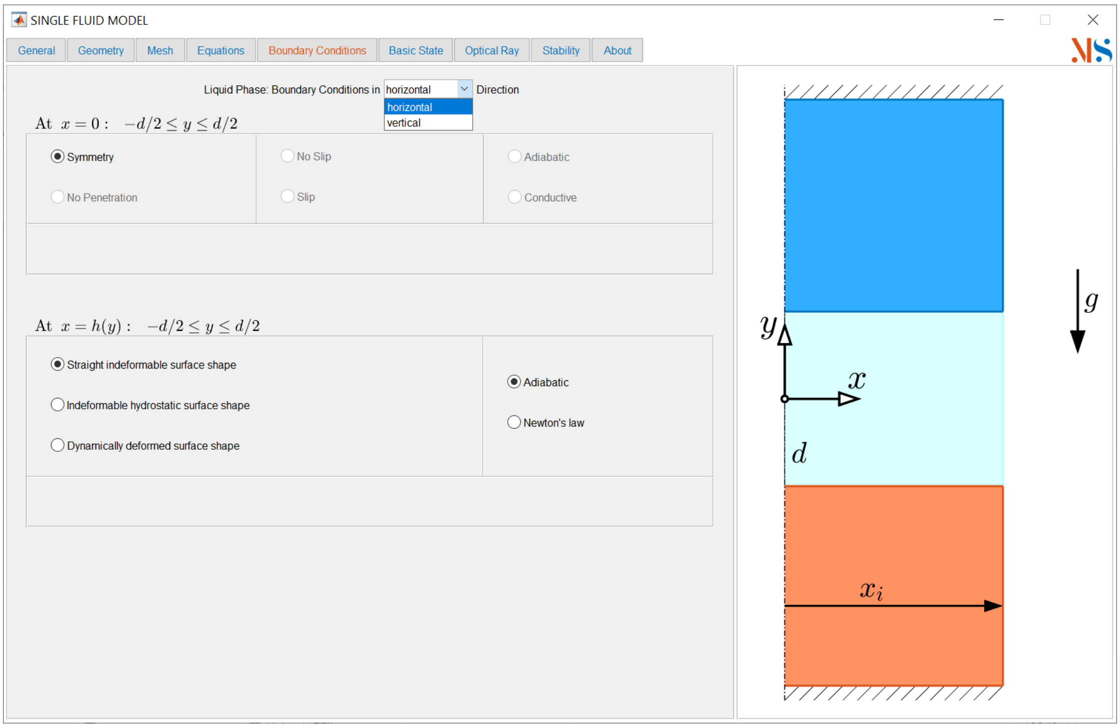


- Leave the energy equation enabled and pay attention to unticking the thermocapillary convection.

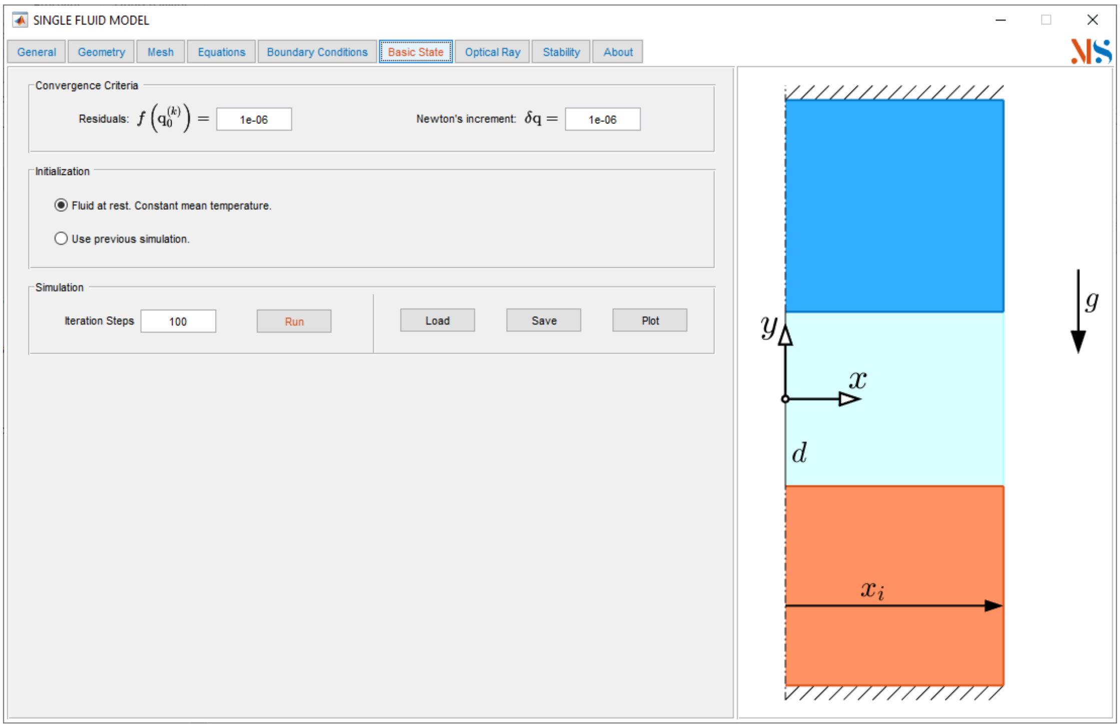


- Set the boundary conditions in the horizontal and vertical directions. Prohibit surface deformations by ticking the box `Straight indeformable surface shape`.

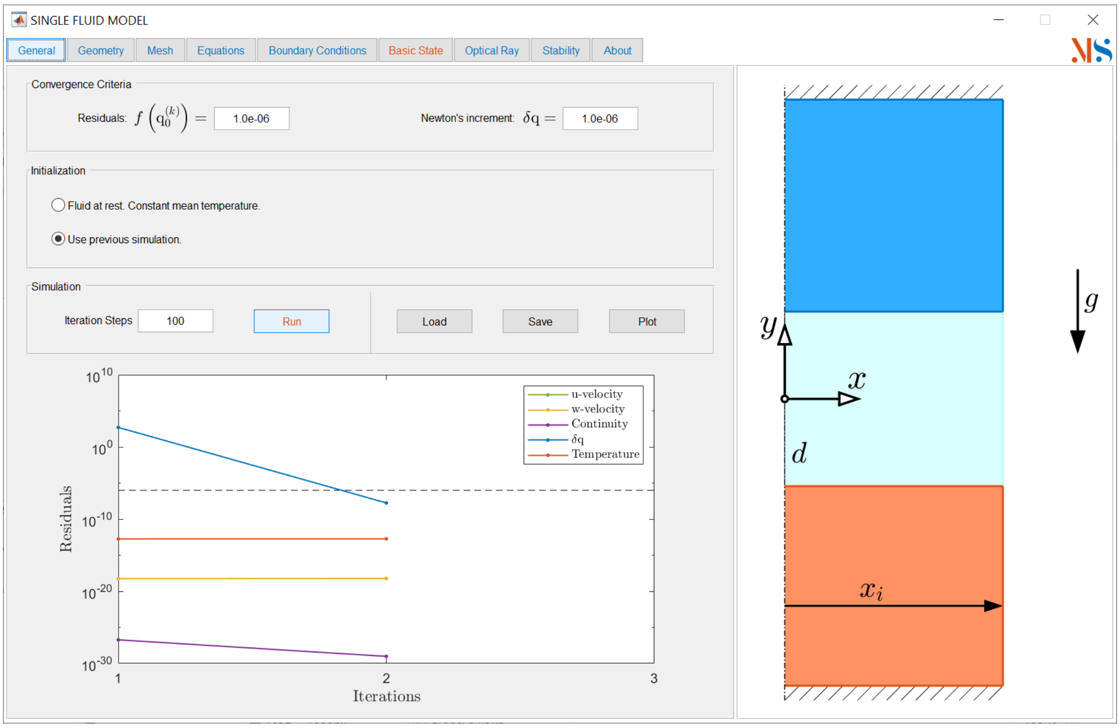




- Set the tolerances of the residuals for the Newton solver and the solution increments.

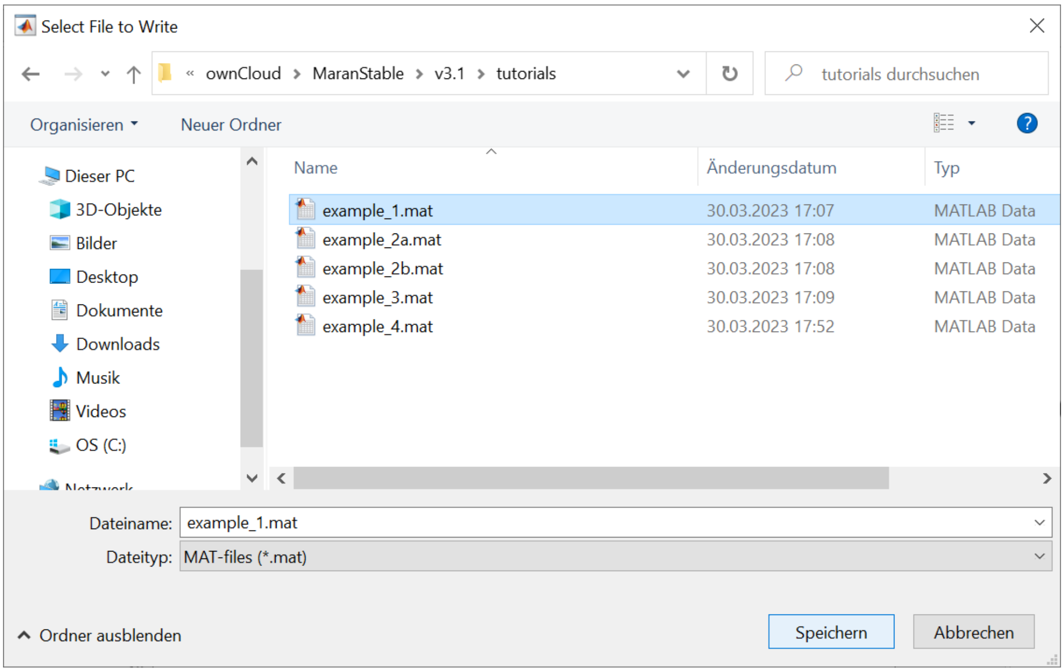


- Solve the basic state by clicking on Run.

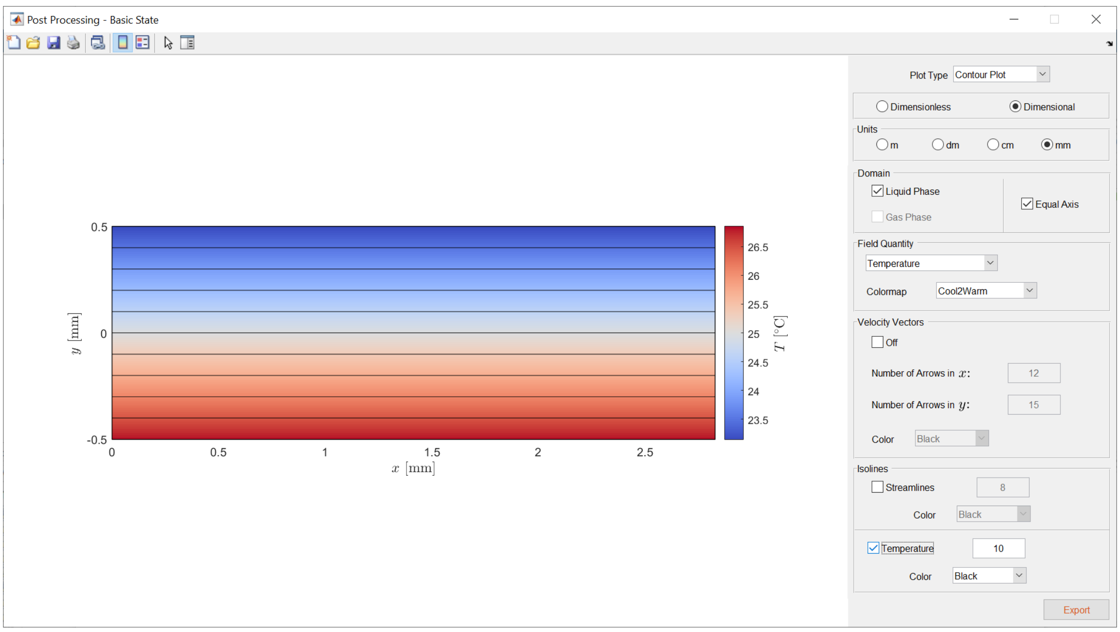


- Upon convergence, `MaranStable` will provide the message:

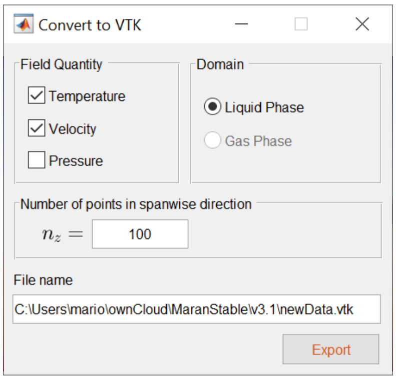
- Click on `Save` to store the converged basic state just computed.



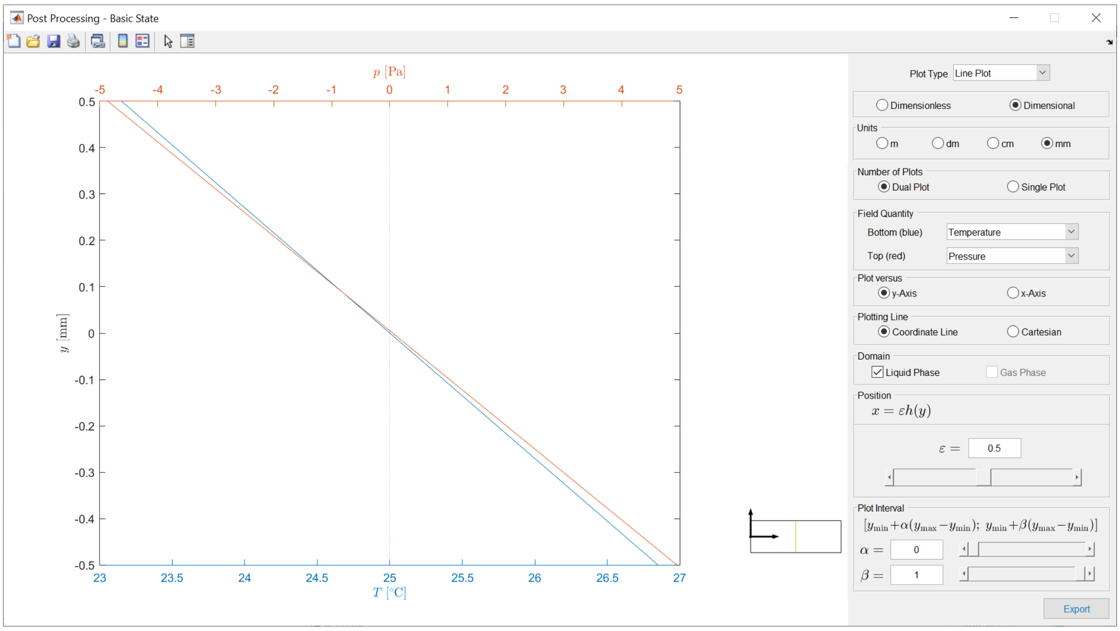
- Click on `Plot` to visualize the basic state. Choose the desired field quantity and enable or disable velocity vectors and/or isolines.



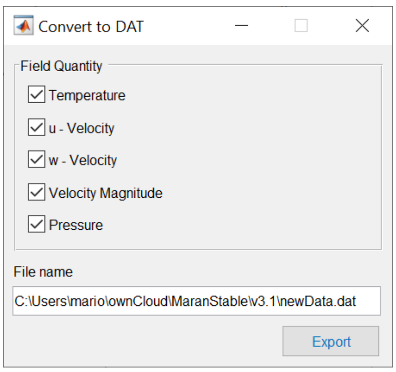
- Click on `Export` to export the solution in vtk format (compatible with ParaView).



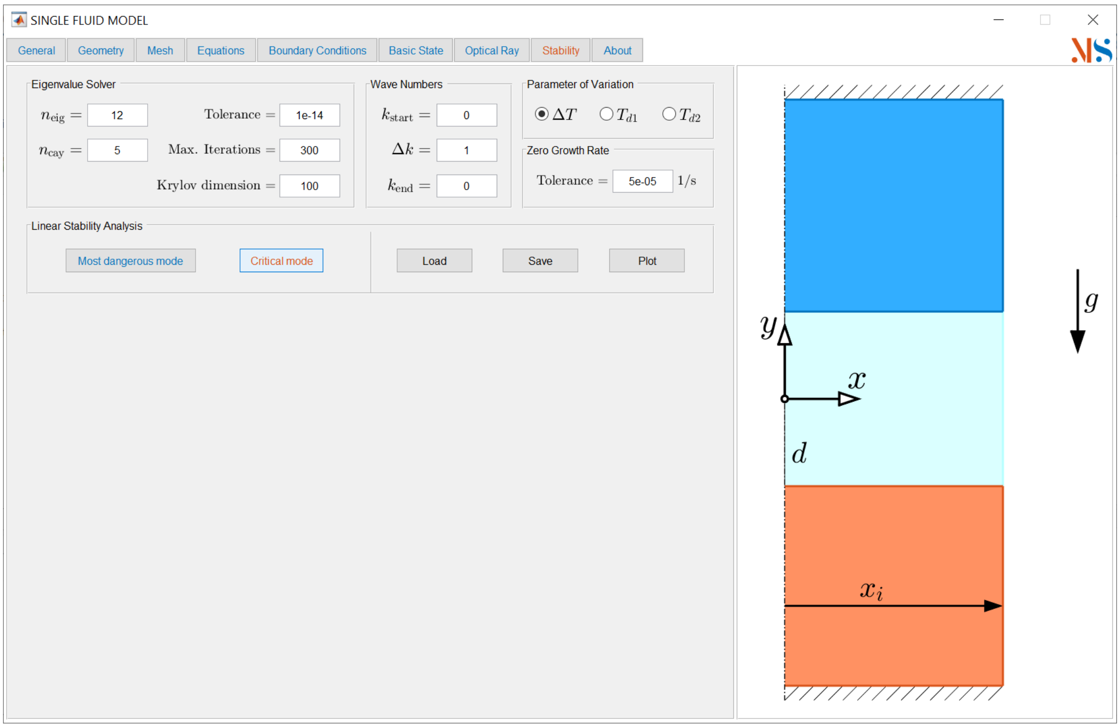
- Change the plot type to `Line Plot` and change the position (location at which the data are evaluated, yellow line) to $\varepsilon = 0.5$ to visualize the temperature and pressure at the vertical midplane.



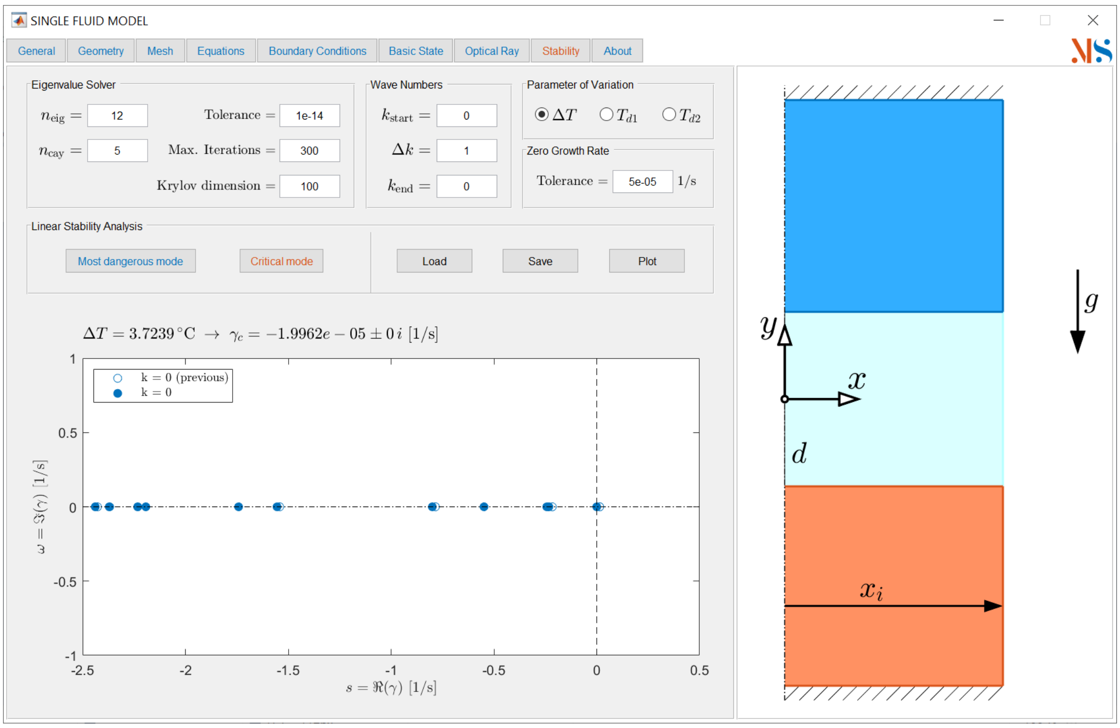
- Click on `Export` to export the solution in dat format (compatible with Xmgrace, Matlab, Excel, etc.).



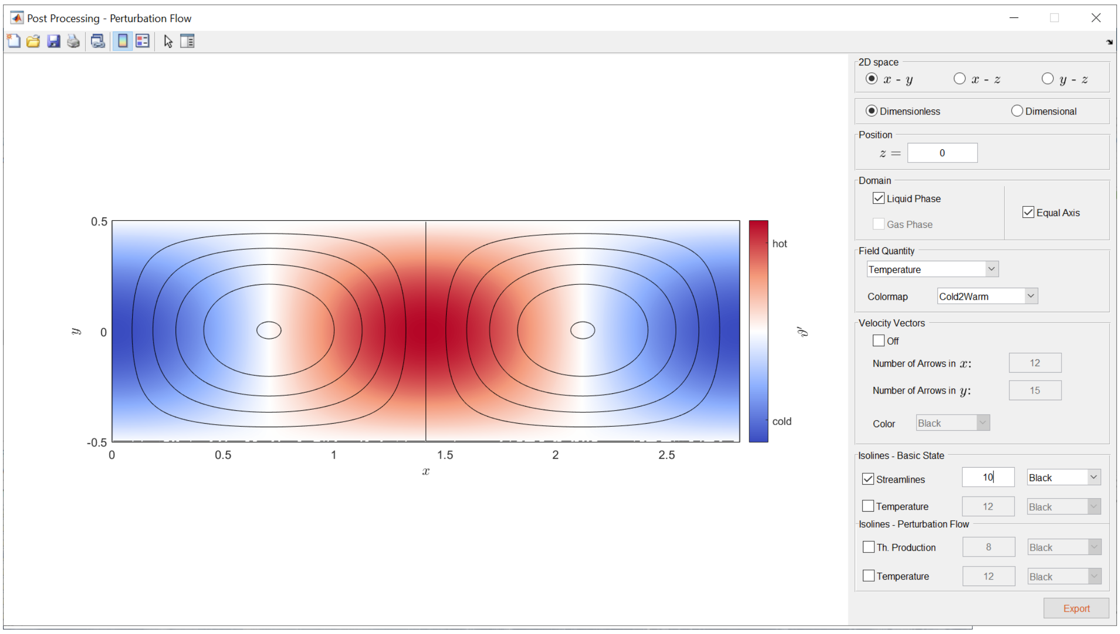
- Set up the parameters for carrying out the stability analysis. The wave number k corresponds to the homogeneous z-direction. Note: Here, this is a different direction from the wavelength λ_c above, which was in the y-direction.



- Solve the linear stability analysis by clicking on `Critical Mode`. Here it arises for $k = 0$ which corresponds to a single pair (one wavelength in y direction selected above) of straight convection rolls which are infinitely extended in the z direction.



- Click on `Plot` to visualize the most critical mode

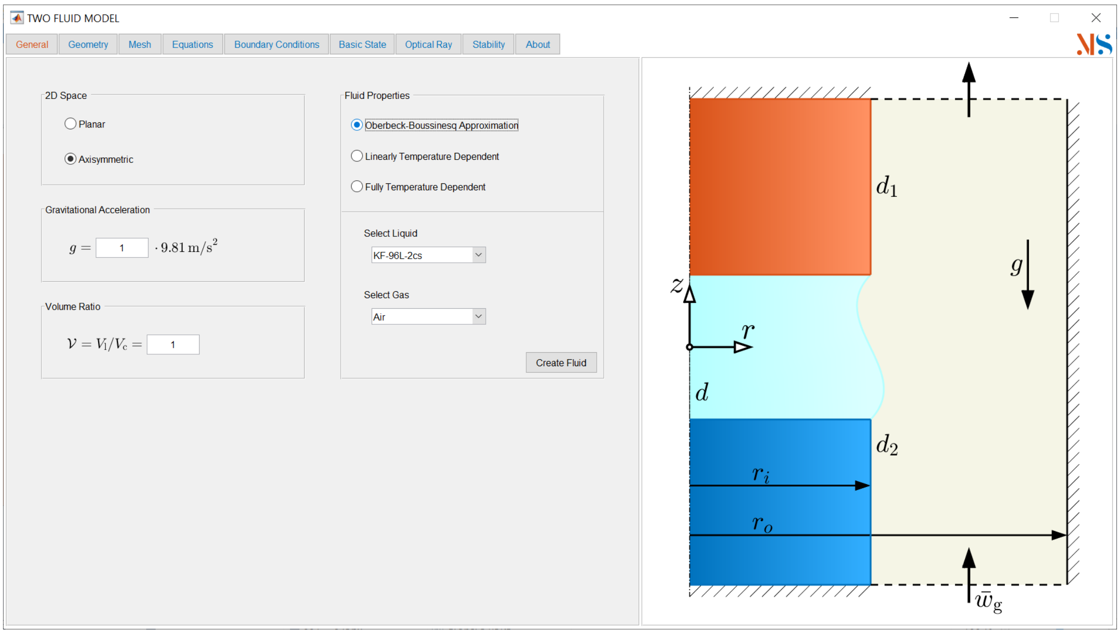


2 Instability of thermocapillary flow

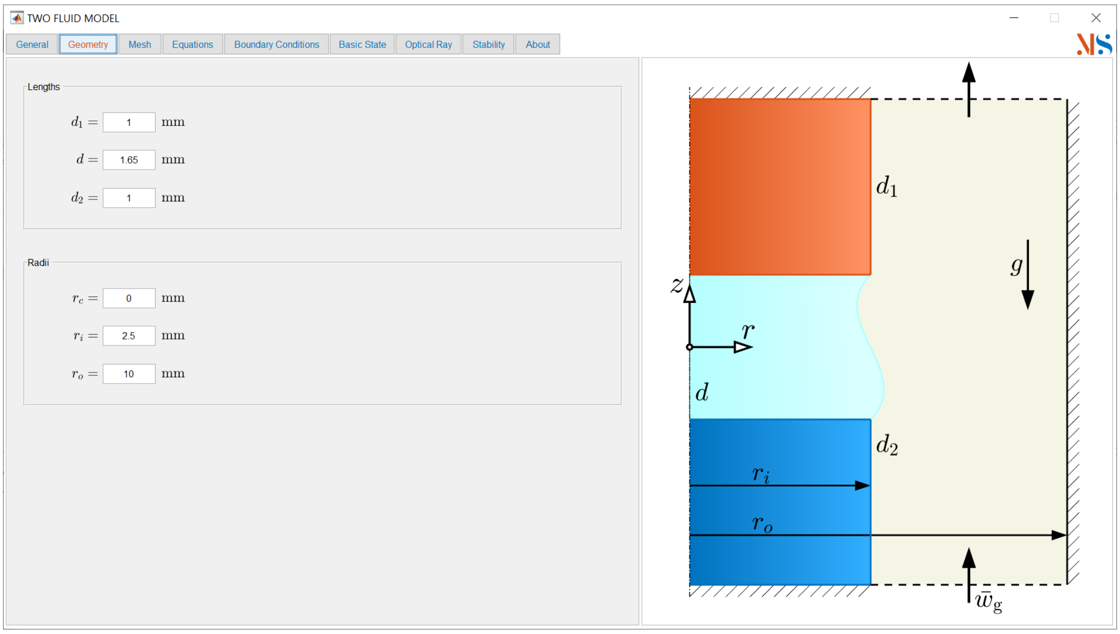
The second tutorial deals with thermocapillary instability in a cylindrical droplet of silicone oil surrounded by air. Both, the liquid and the gas are assumed to be Boussinesq fluids. This tutorial corresponds to the file `example_2a.mat` in the folder `tutorials`. Its counterpart with fully temperature-dependent properties is stored in the file `example_2b.mat`.

- Select the two-phase solver

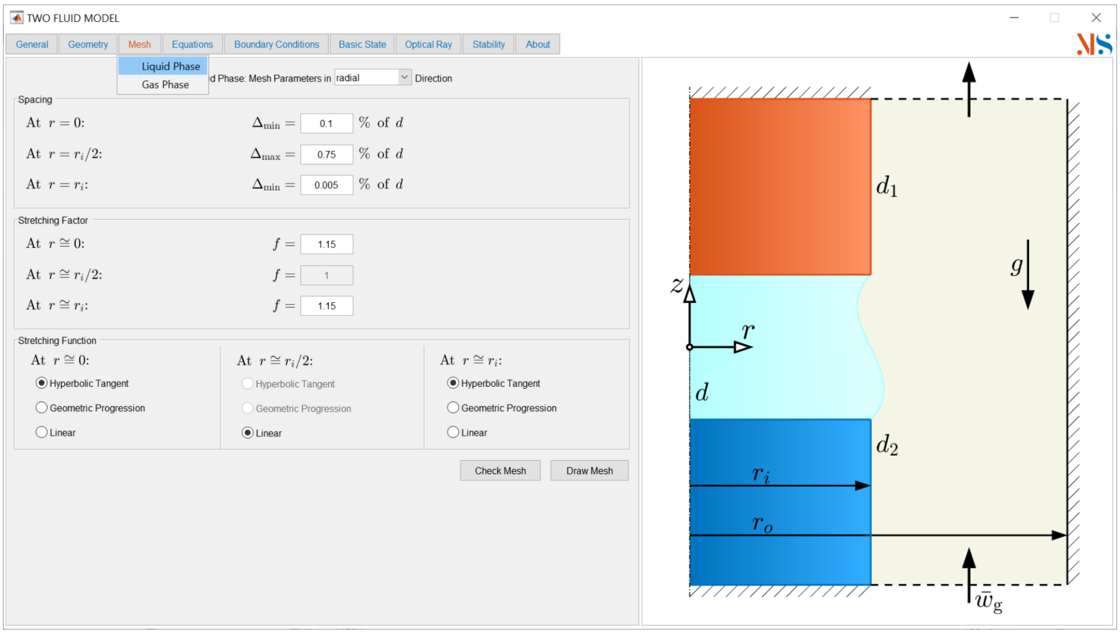
- Set up the axisymmetric geometry and the gravity acceleration. The volume ratio is set to 1 and a 2cSt-silicone oil is considered. Select the Oberbeck–Boussinesq approximation

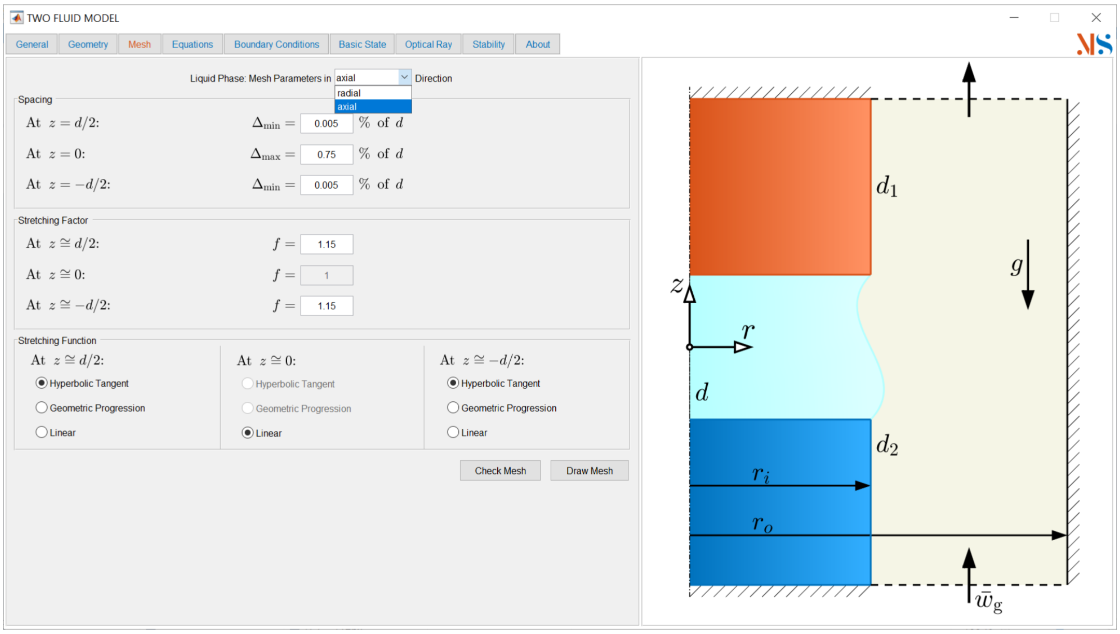


- Set up the cylindrical geometry as follows. The values are selected to coincide with the ones reported in Stojanović et al. (n.d.).

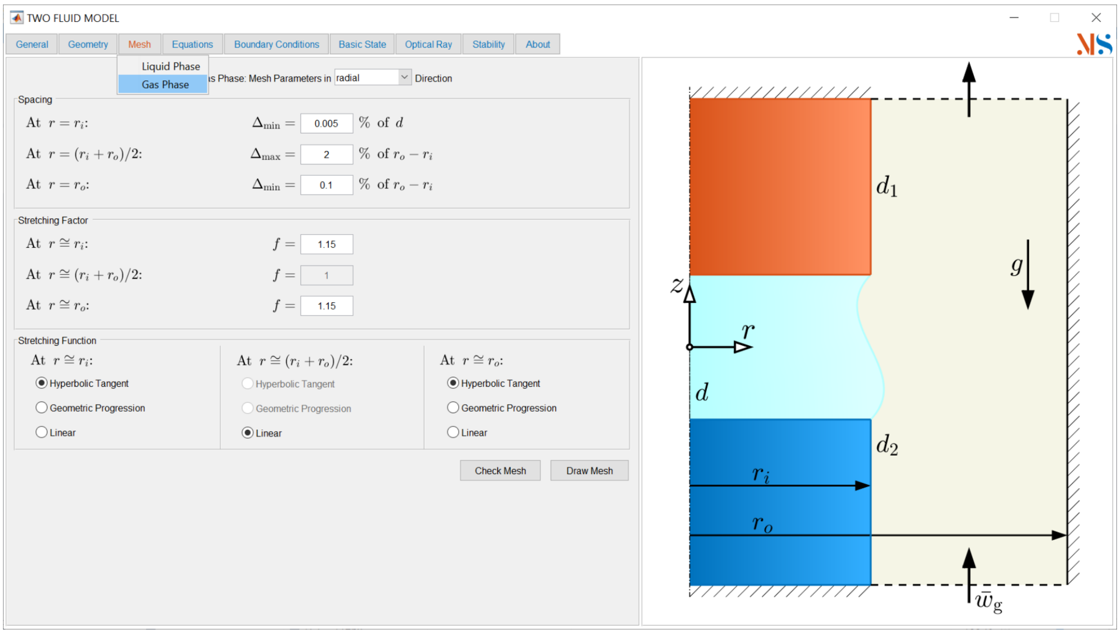


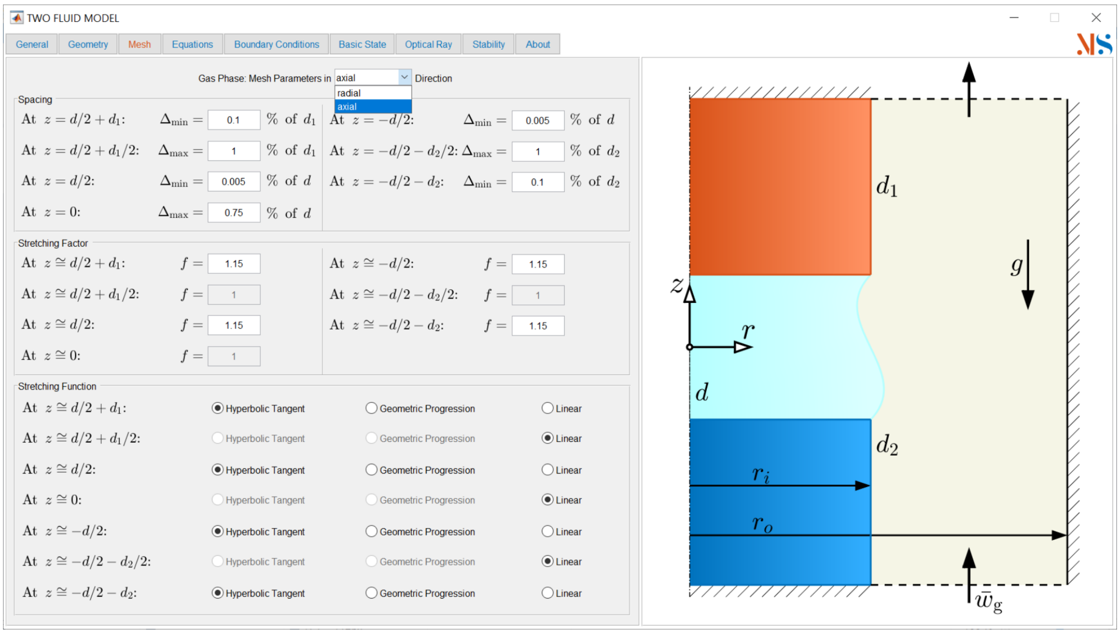
- Set up the mesh for the liquid phase by clicking on `Mesh` and `Liquid Phase`. Define the parameters for both radial and axial coordinates.



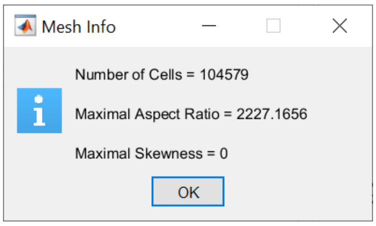


- Set up the parameters for the gas phase in an analogous manner for both radial and axial coordinates.

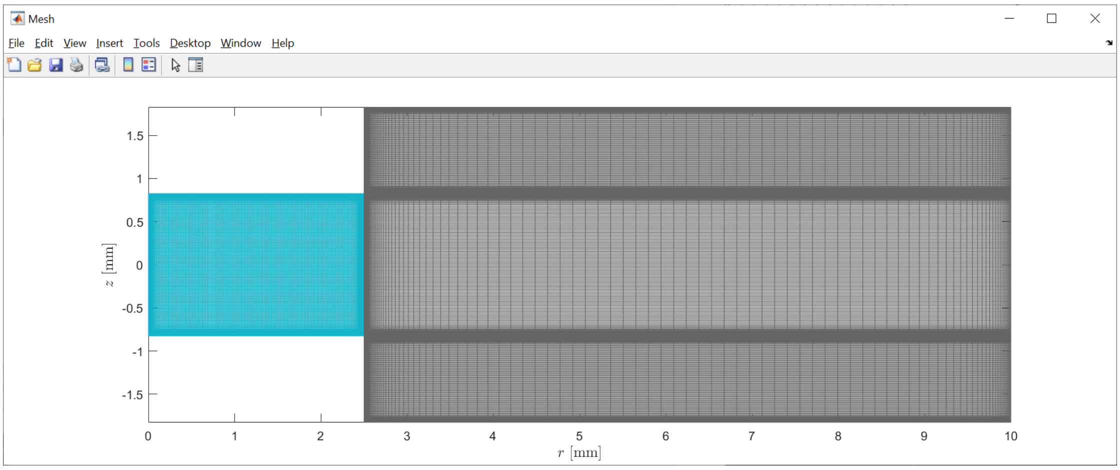




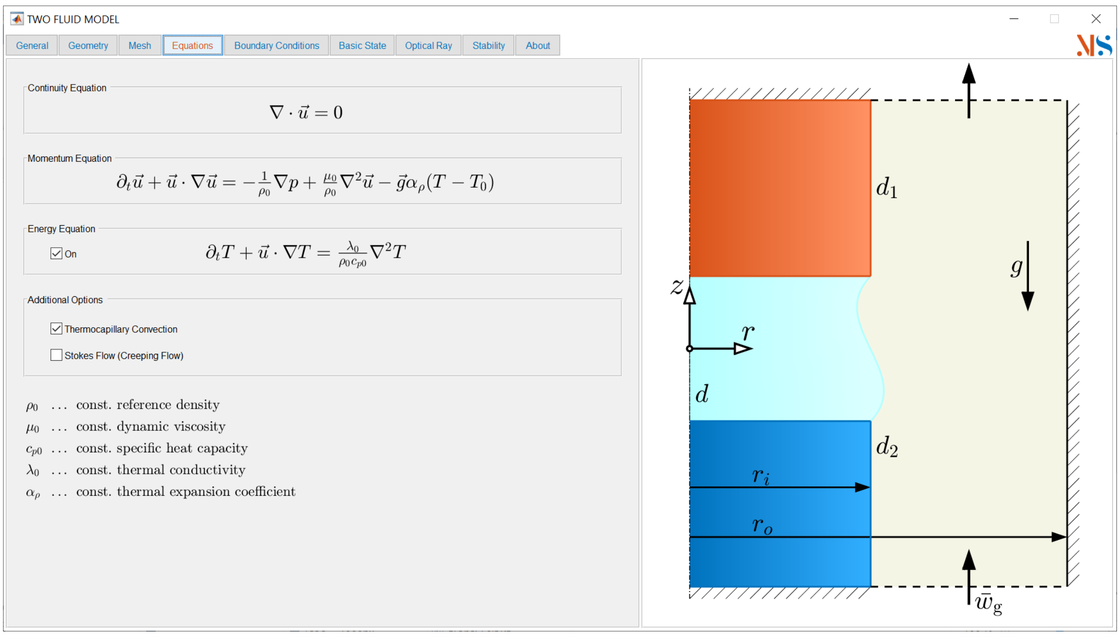
- Click on `Check Mesh`



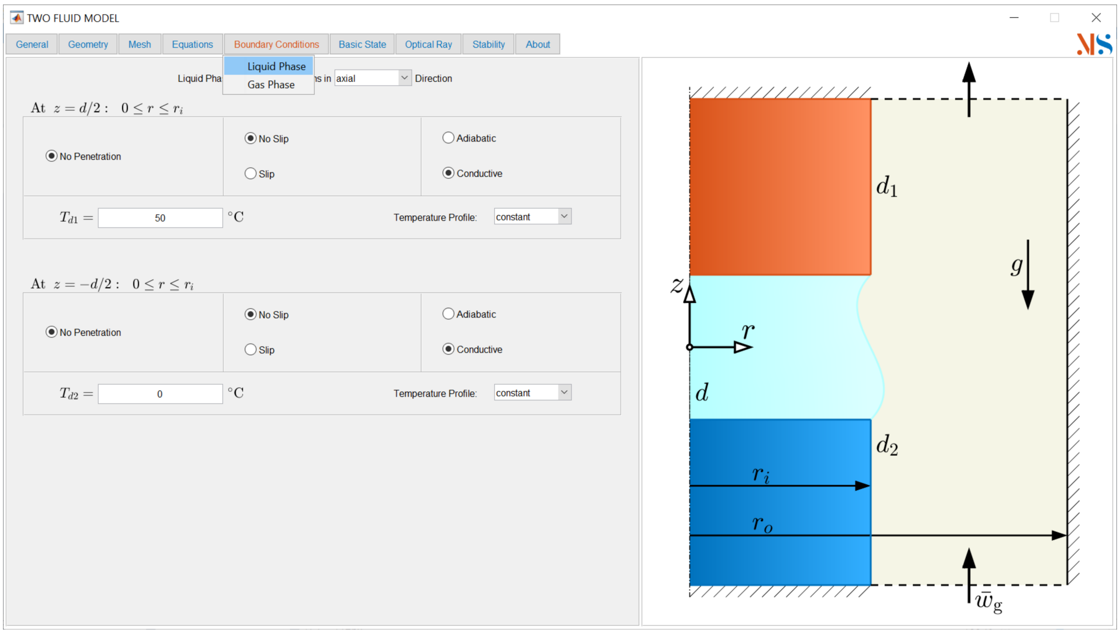
- Visualize the mesh by clicking `Draw Mesh`

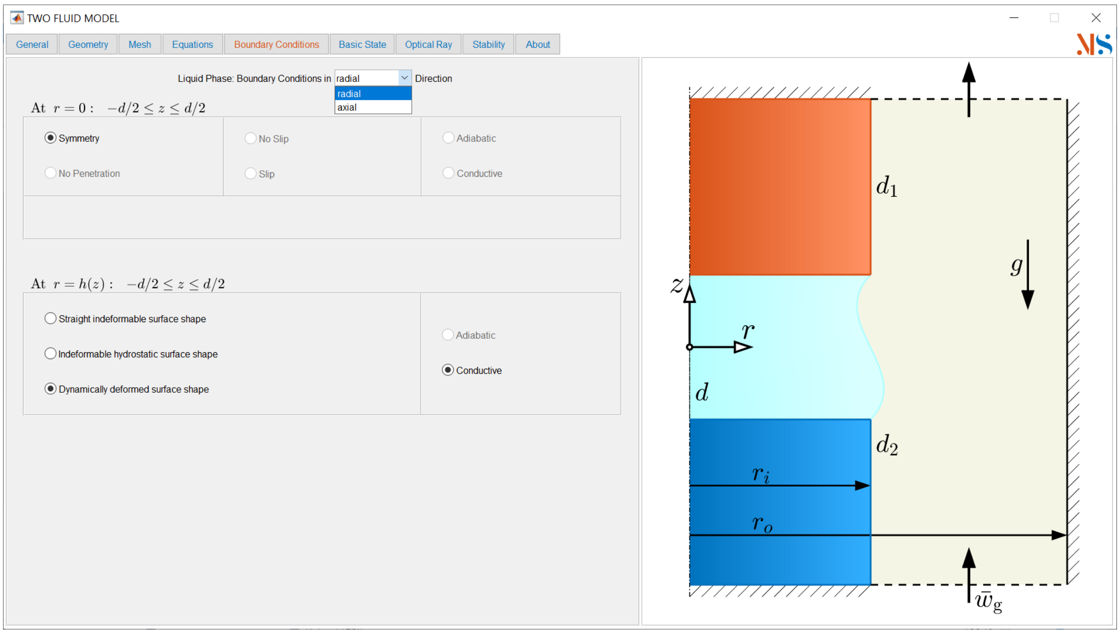


- Select the equations to solve by keeping active the energy equation and the thermocapillary convection.

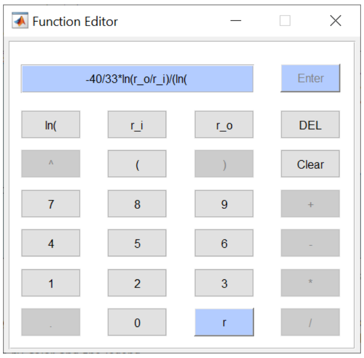


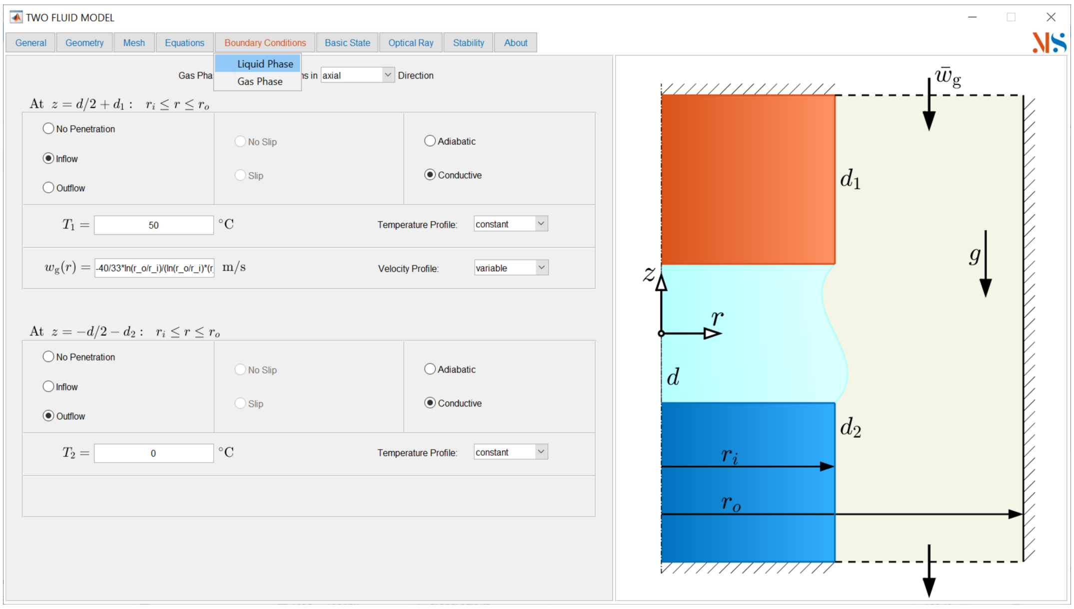
- Set the boundary conditions in the radial and axial directions for the liquid phase. Pay attention to including dynamic surface deformation.

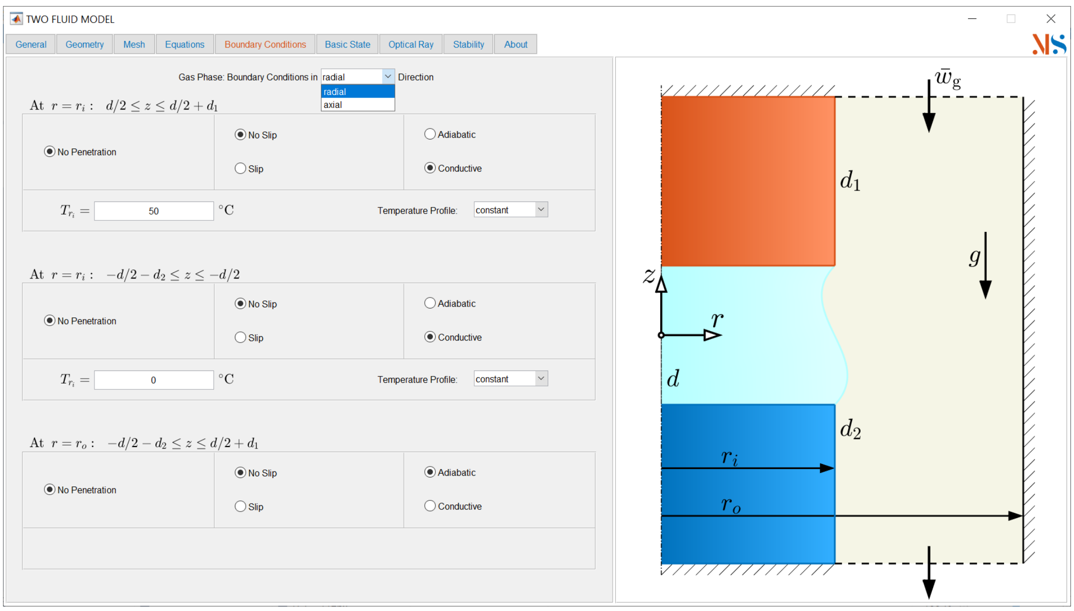




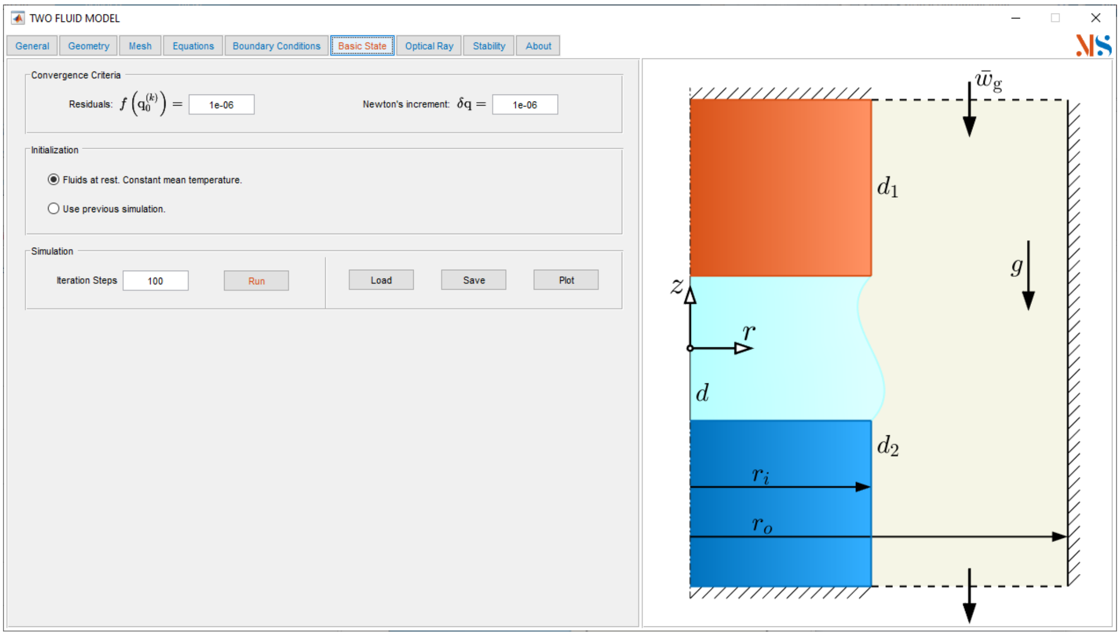
- Set the boundary conditions in the radial and axial directions for the gas phase. Select `variable` in the popup menu for the velocity profile. Use the function editor to prescribe a fully developed profile for $w_\mathrm{g}(r)$. The implemented profile is fully developed with a mean inlet velocity of $\bar{w}_\mathrm{g} = -(20/33)\,\mathrm{m/s}$ (Joseph, 1976).



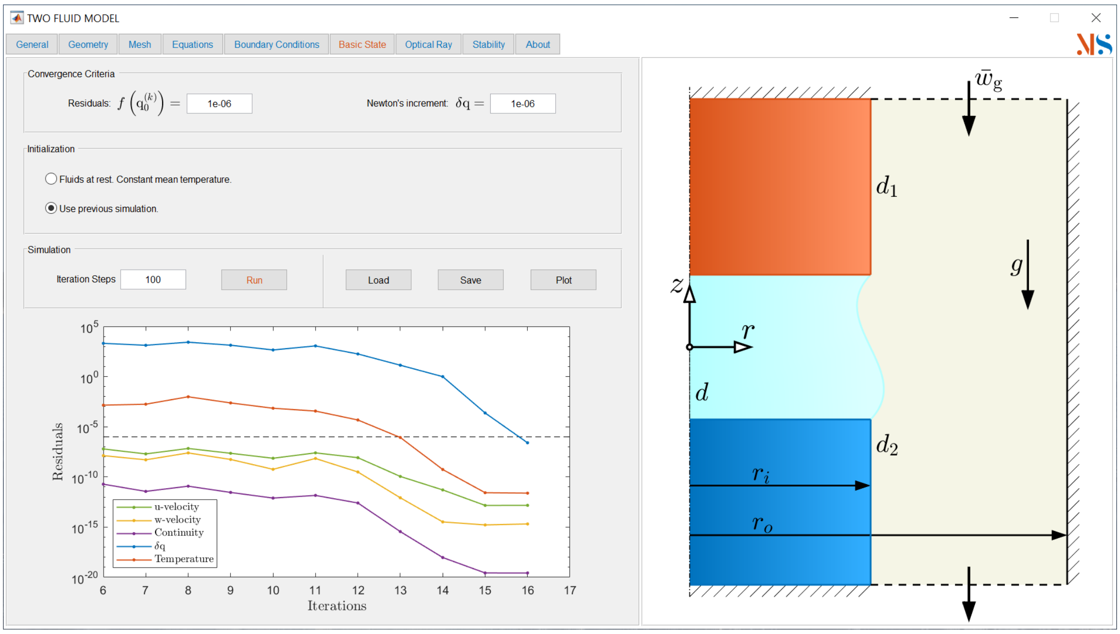




- Set the tolerances of the residuals for the Newton solver and the solution increments.

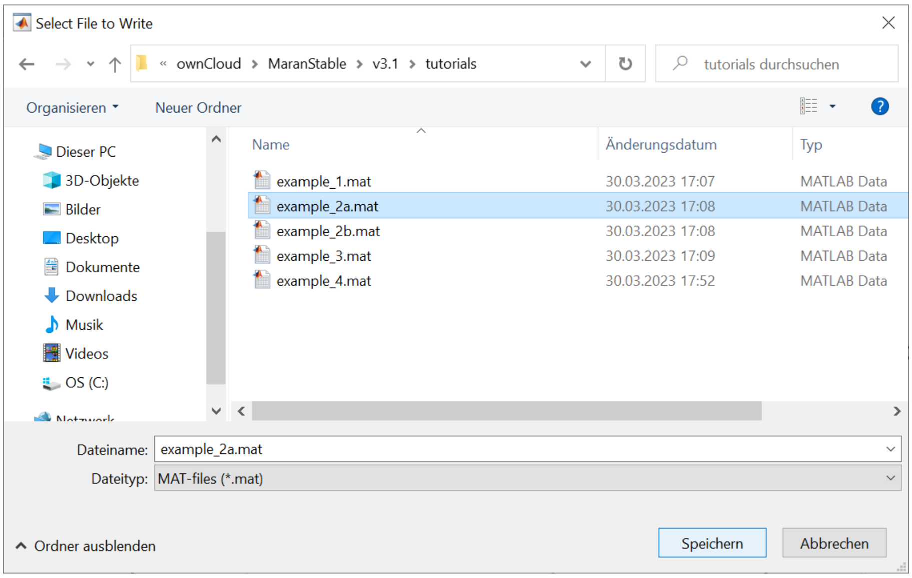


- Solve the basic state by clicking on `Run`.

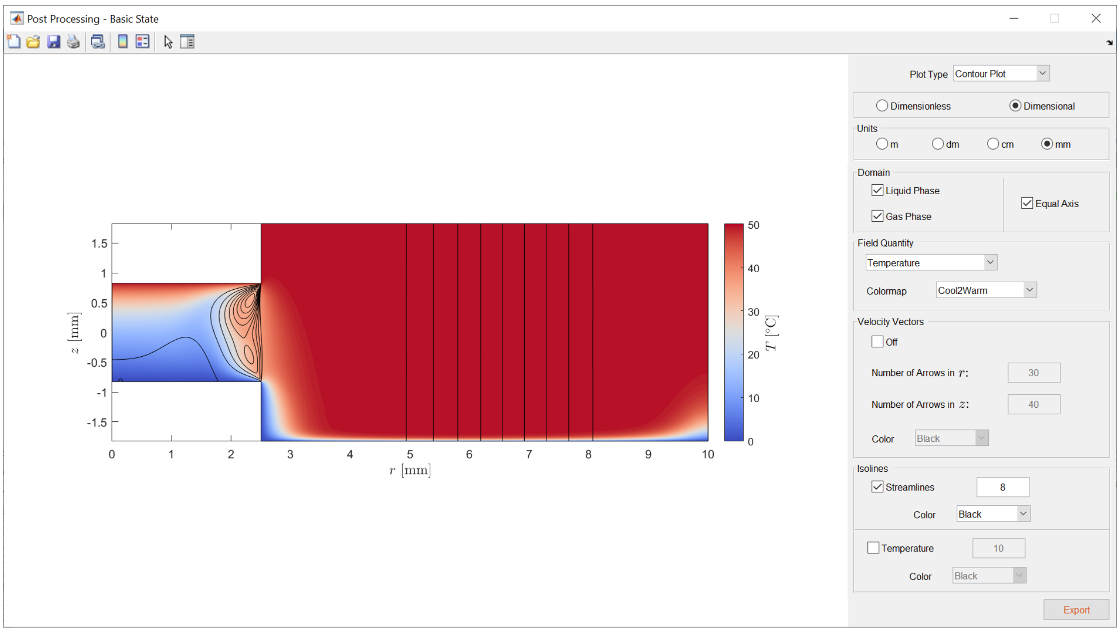


- Upon convergence, `MaranStable` will provide the message:

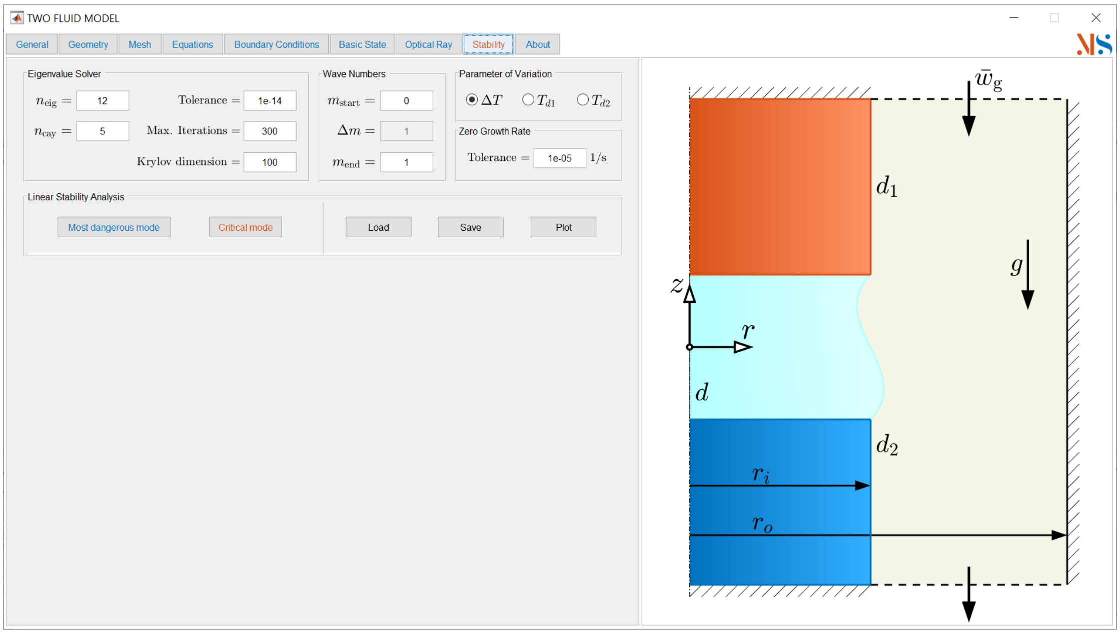
- Click on `Save` to store the converged basic state just computed.



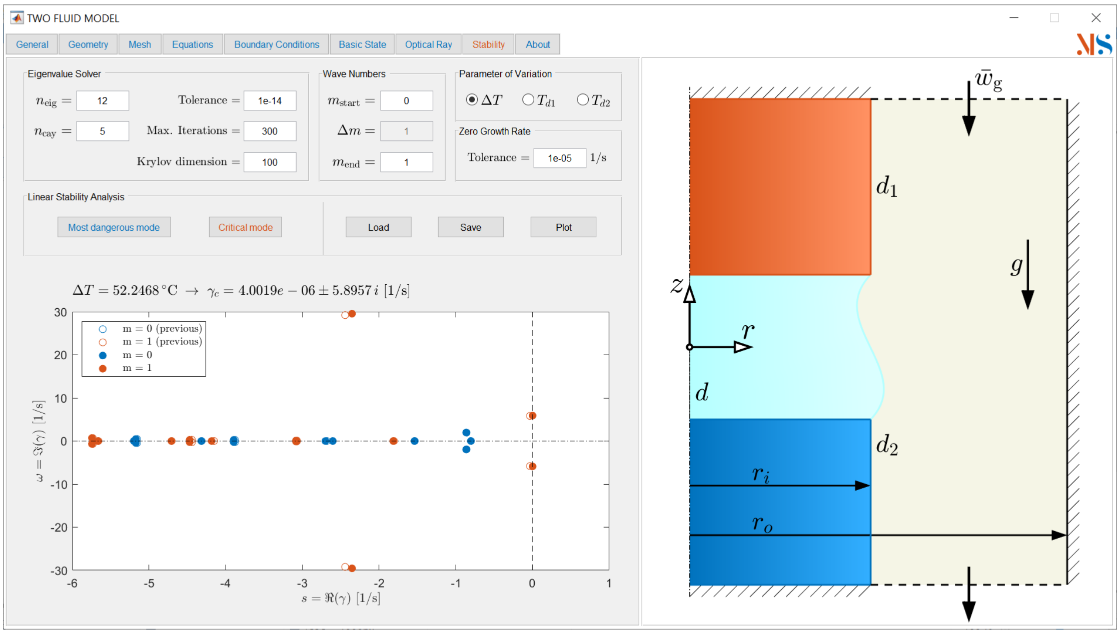
- Click on `Plot` to visualize the basic state.



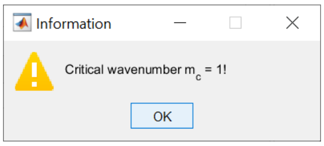
- Set up the parameters for carrying out the stability analysis. The wave number m corresponds to the homogeneous φ-direction.



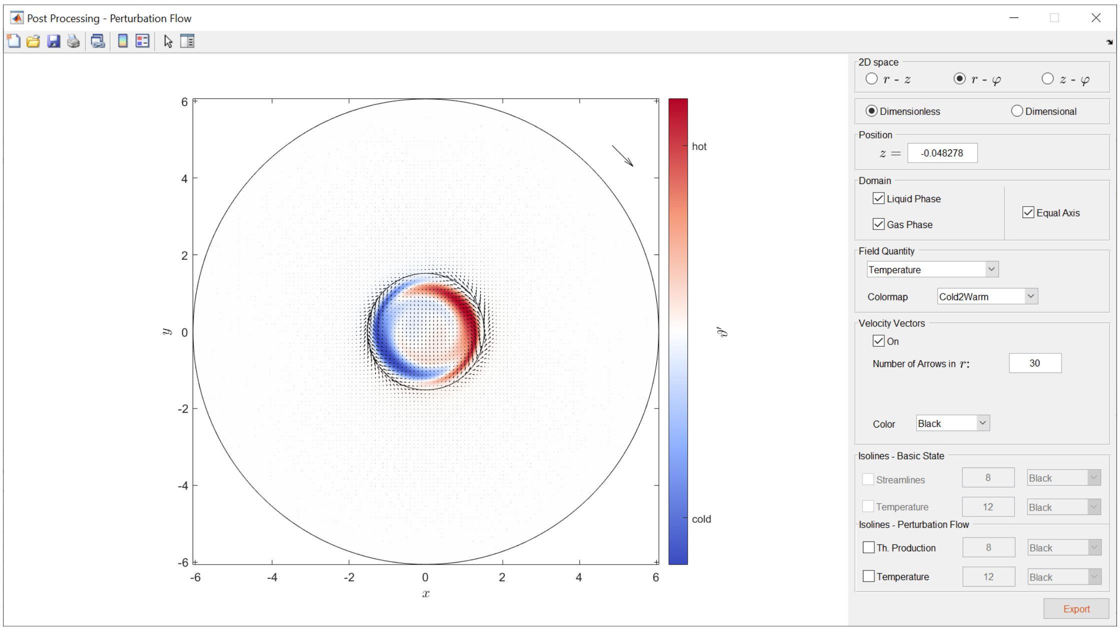
- Solve the linear stability analysis by clicking on `Critical Mode`.



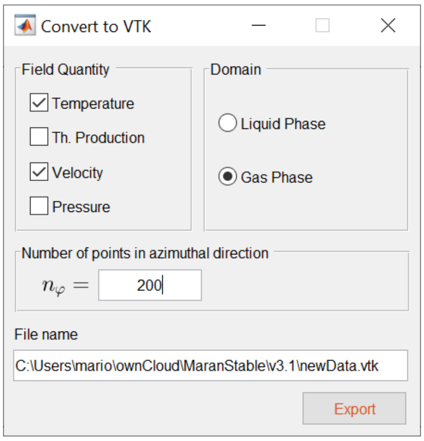
- Once the linear stability analysis is finished, `MaranStable` will provide the message:



- Click on `Plot` to visualize the most critical mode and select $r - \varphi$ in the 2D-space box.



- Export the critical mode in vtk format by clicking on `Export`. Select the field quantities to be exported and the domain (liquid or gas). Increase the number of nodes in the azimuthal direction, if necessary.

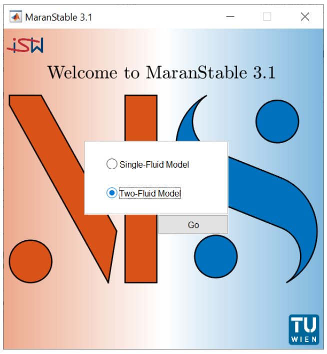


Please note that for the counterpart `example_2b.mat`, computing the basic state for $\Delta T = 50\,\mathrm{K}$ with the initialization 'Fluids at rest. Constant mean temperature.' will run into an error because of the higher-order nonlinearity of the FTD equations. To circumvent this problem, first compute the basic state with e.g. $\Delta T = 30\,\mathrm{K}$ ($T_{\mathrm{hot}} = 40°\mathrm{C}$, $T_{\mathrm{cold}} = 10°\mathrm{C}$). After the basic state has been computed, change the boundary conditions to $\Delta T = 50\,\mathrm{K}$ and use the just computed flow field as initialization.

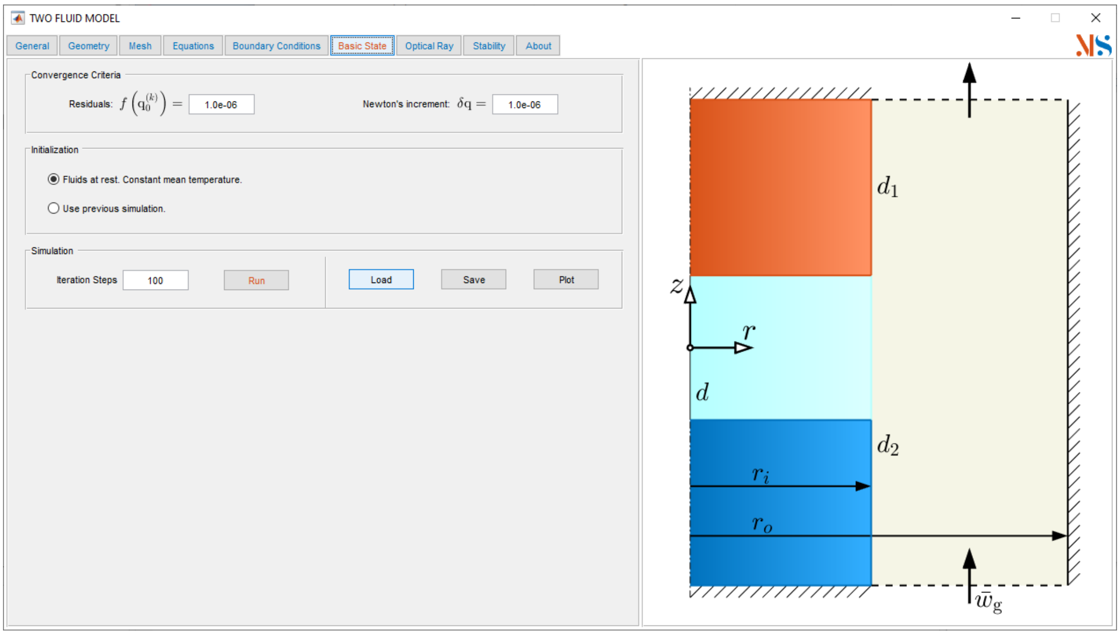
3 Optical ray tracing

The third tutorial deals with tracing an optical ray in a temperature-dependent index of refraction $\mathcal{N}(T)$. This tutorial corresponds to the file `example_3.mat` in the folder `tutorials`.

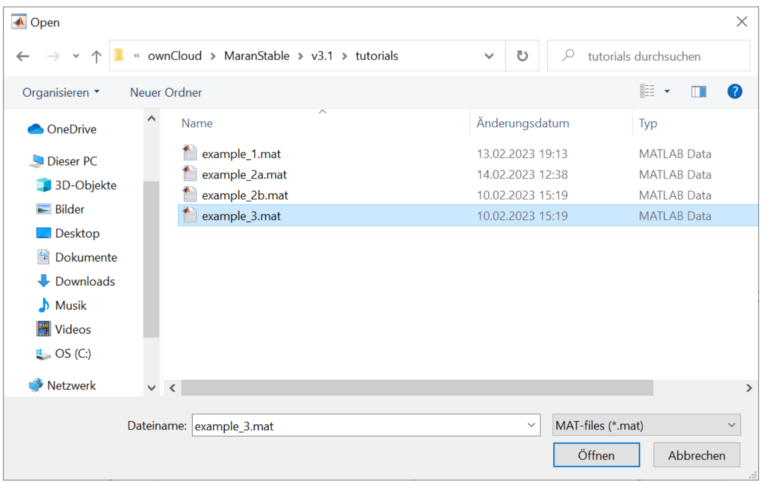
- Select the two-phase solver



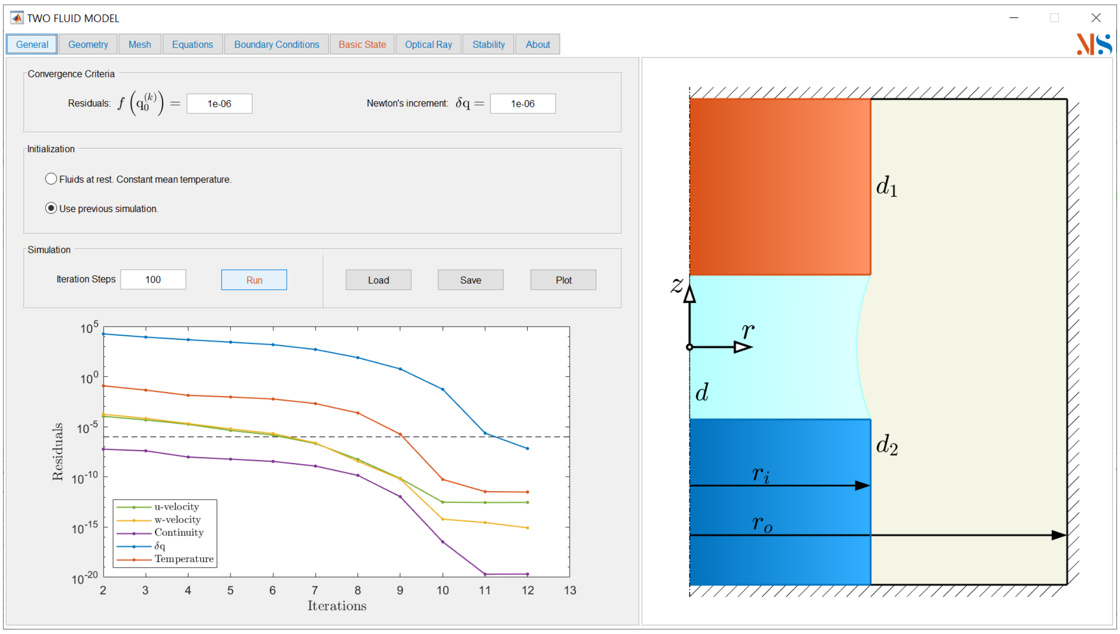
- Select the tab `Basic State`



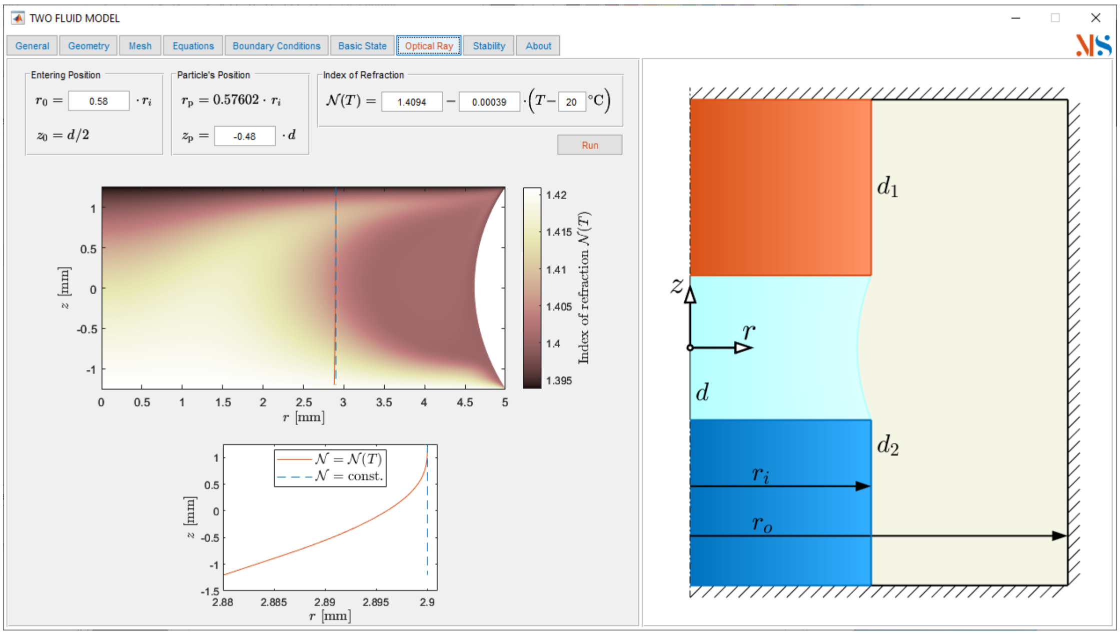
- Load the pre-defined case file.



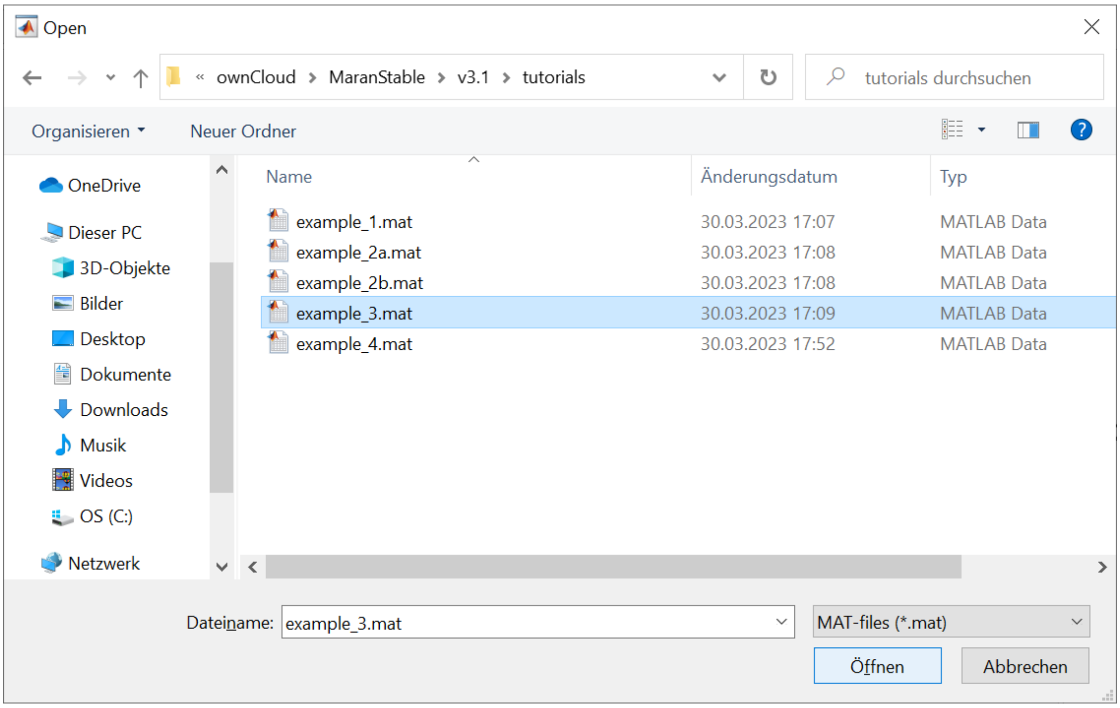
- Click on Run to compute the basic state.



- Set up the ray tracing parameters including the location of the ray entering the domain and the temperature dependence of the refractive index $\mathcal{N}(T)$. Leave the default refractive index $\mathcal{N}(T)$ unchanged if dealing with silicone oils (He et al., 2016).



- Click on `Run` to compute the optical ray tracing.

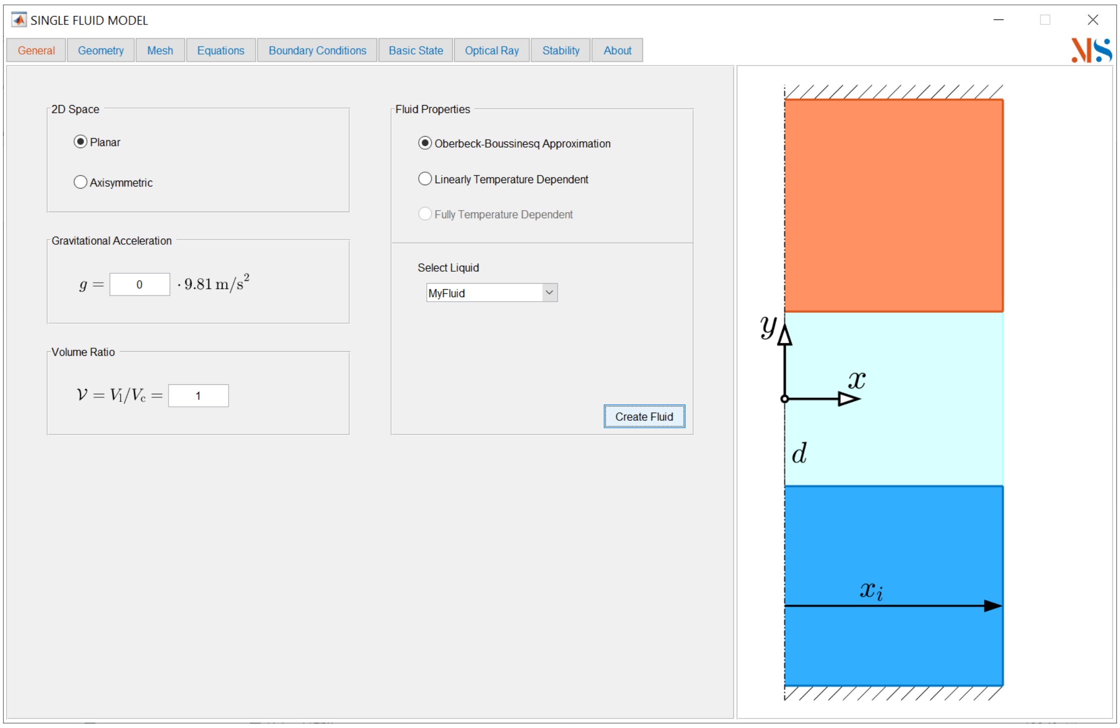


4 Shear-driven cavity

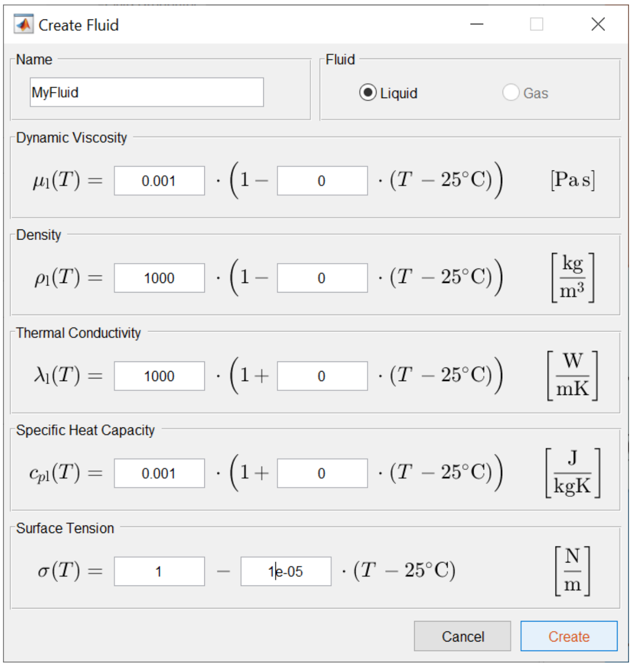
The fourth tutorial deals with a shear-stress-driven cavity. It is aimed to reproduce the steady two-dimensional flow presented in Romanò and Kuhlmann (2017). To realize a constant shear force on the interface, we consider the thermocapillary flow in the limit of vanishing Prandtl number. In this case, the temperature field is not affected by the fluid motion and is determined by conduction only. Since the conducting temperature profile on the interface varies linearly between the two heated walls, the thermocapillary effect leads to a constant stress in the interface. In this example, we select $\mathrm{Pr} = 10^{-9}$ and impose a temperature difference corresponding to $\mathrm{Re} = 1000$ and compute the 2D basic flow. This tutorial corresponds to the file `example_4.mat` in the folder `tutorials`.

- Select the single-phase solver

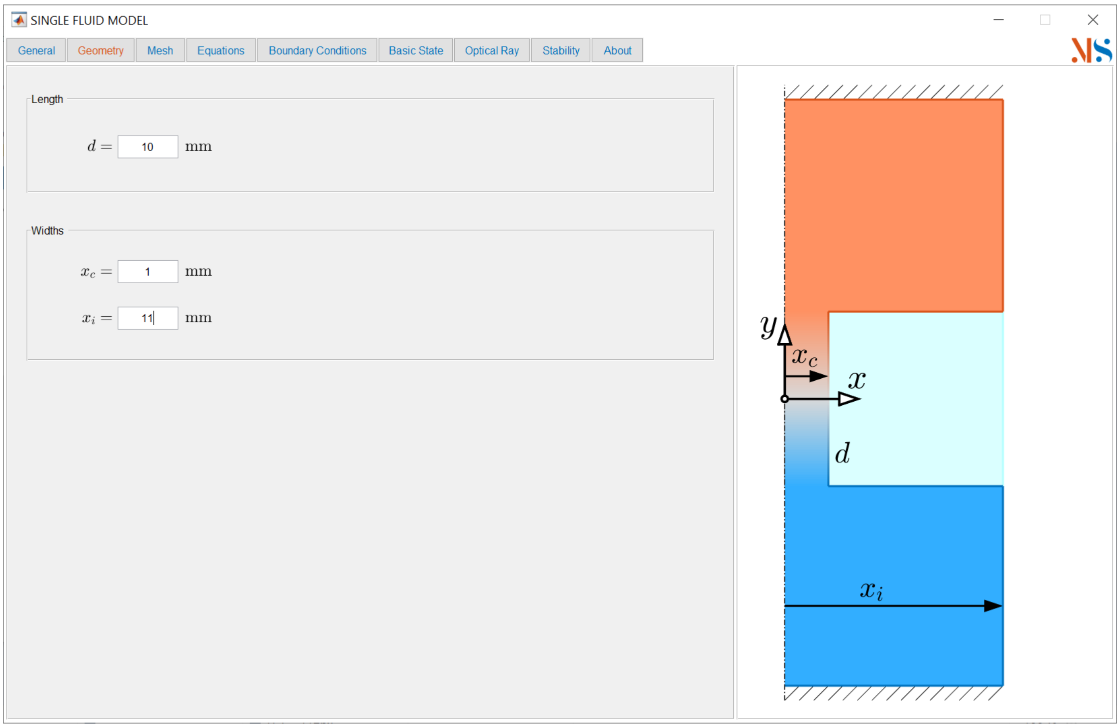
- Select a planar geometry and keep the zero-gravity condition. Since we consider a rectangular domain, the volume ratio is set to $\mathcal{V} = 1$. Select the Oberbeck–Boussinesq approximation as physical model.



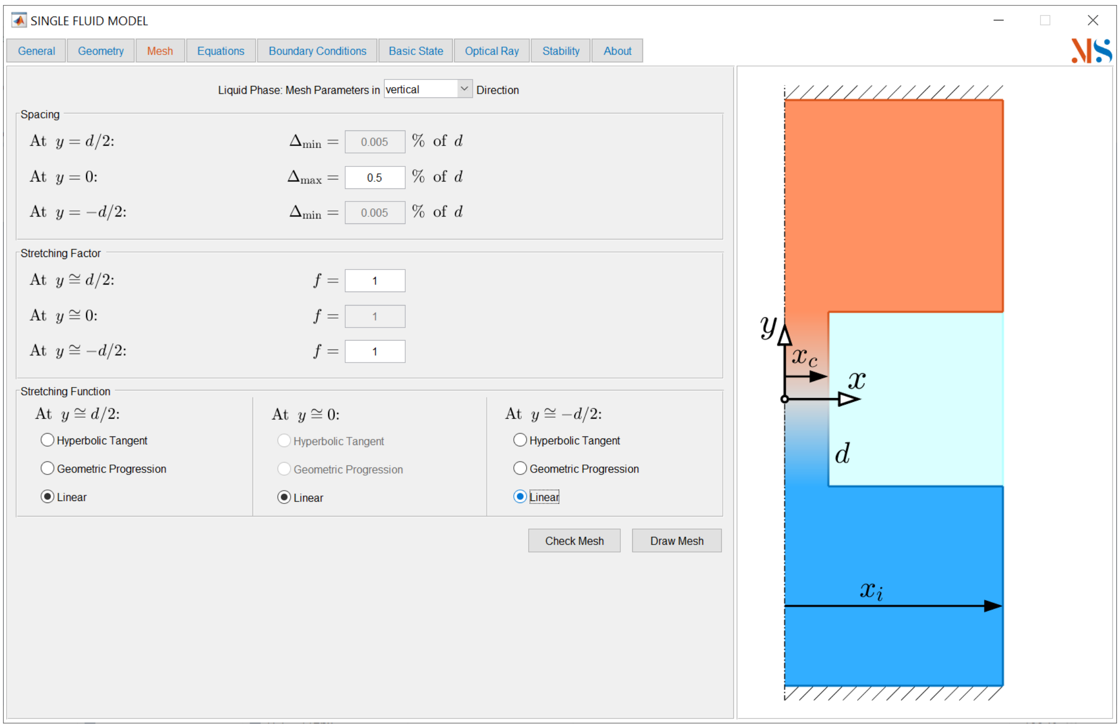
- Set up a new fluid by clicking on `Create Fluid`. The parameters are selected to obtain a fluid with $\mathrm{Pr} = \mu_l c_{pl}/\lambda_l = 10^{-9}$. Then click `Create`.



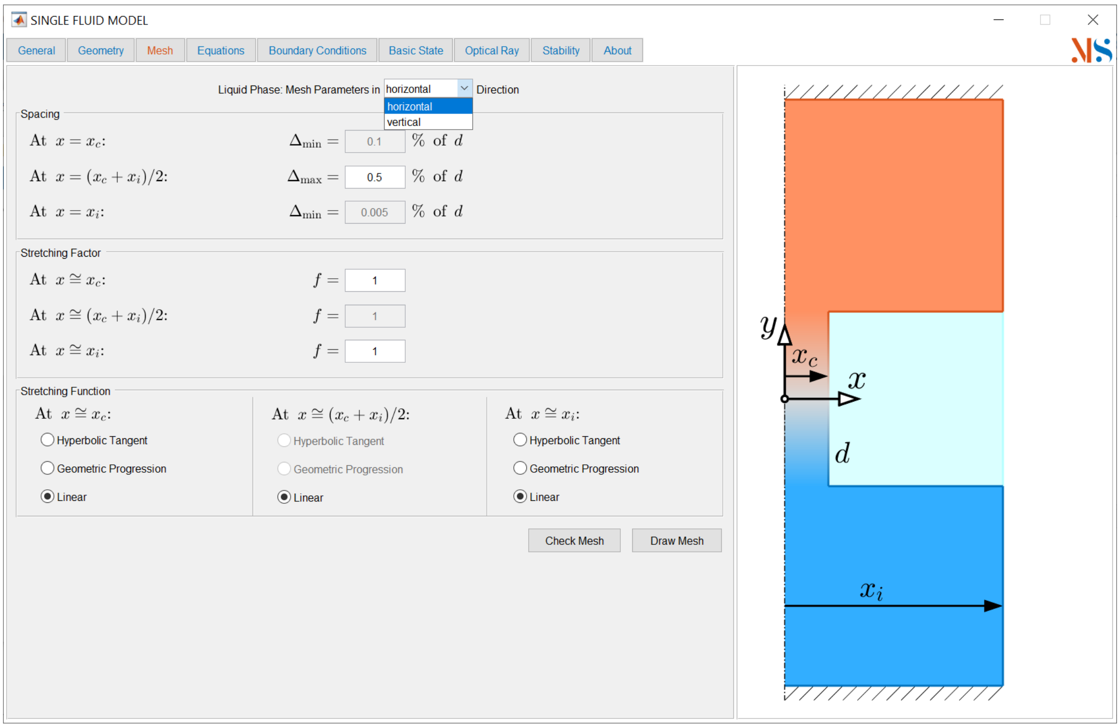
- Set up the planar geometry as follows. Select a non-zero value for x_c to turn the liquid bridge into a cavity. Then specify d and x_i to obtain a quadratic cavity, i.e. $d = x_i - x_c$.



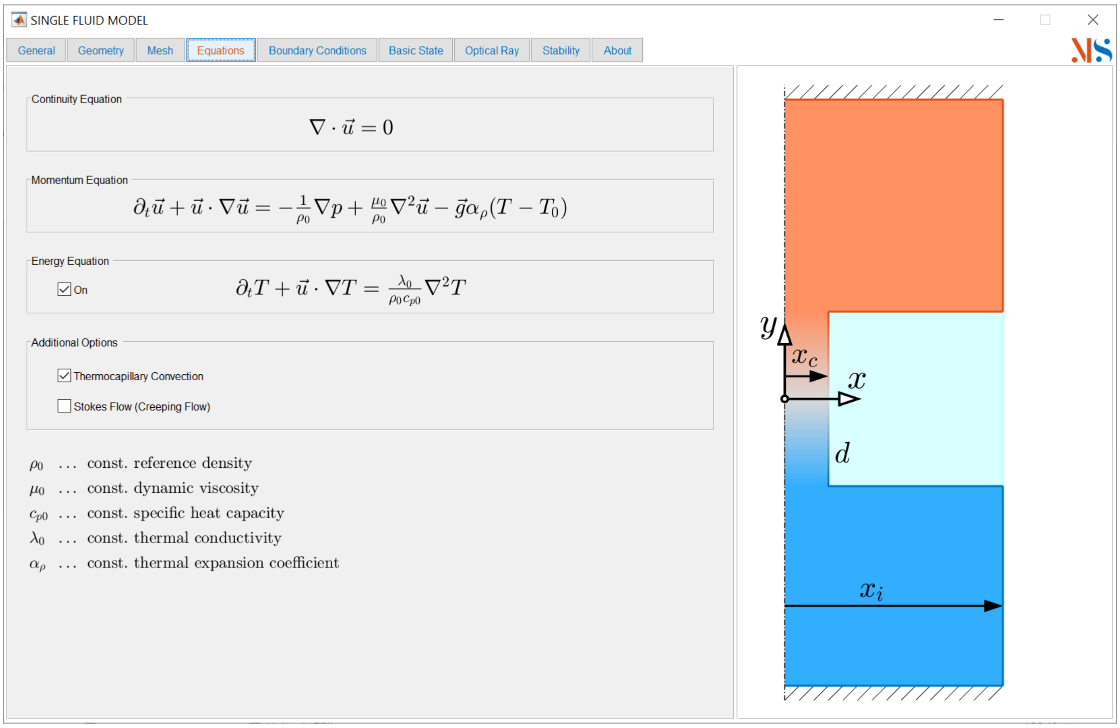
- Set up the mesh parameters for the vertical coordinate y as follows. Choose an equidistant distribution of the cells (stretching factor 1) with a cell width of 0.5% of the total height.



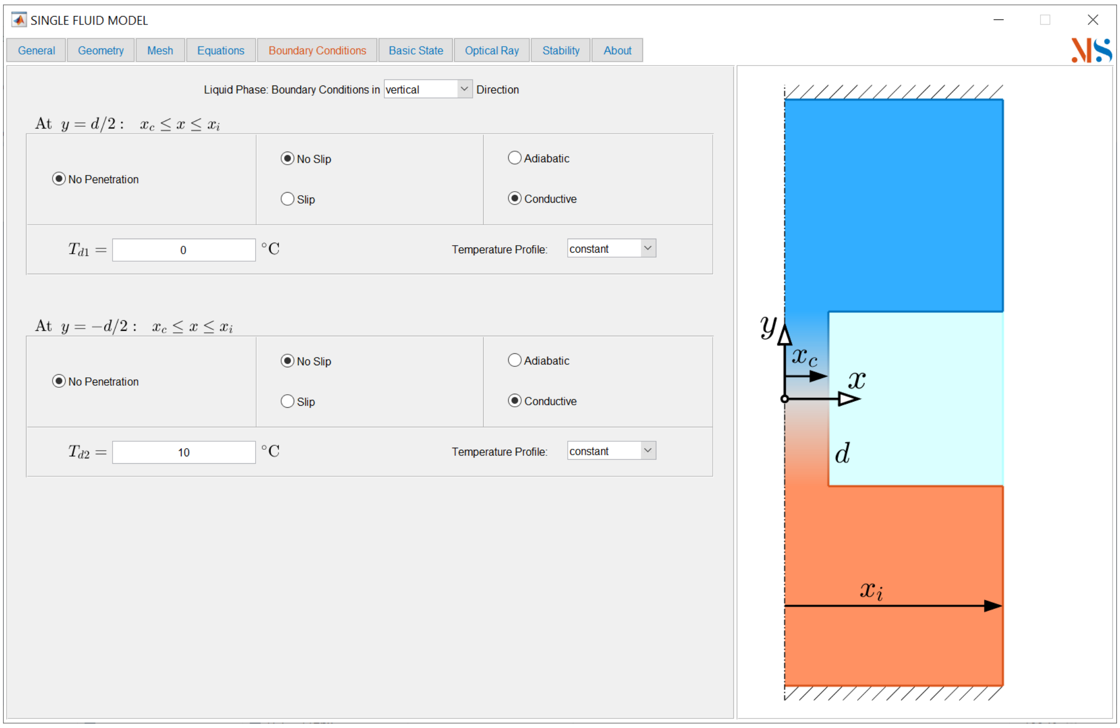
- Repeat the procedure for the horizontal coordinate x. A cell width of $0.005d$ fills the domain with $200 \times 200 = 40\,000$ cells. You may check and/or plot the mesh using **Check Mesh** and/or **Draw Mesh**, respectively.



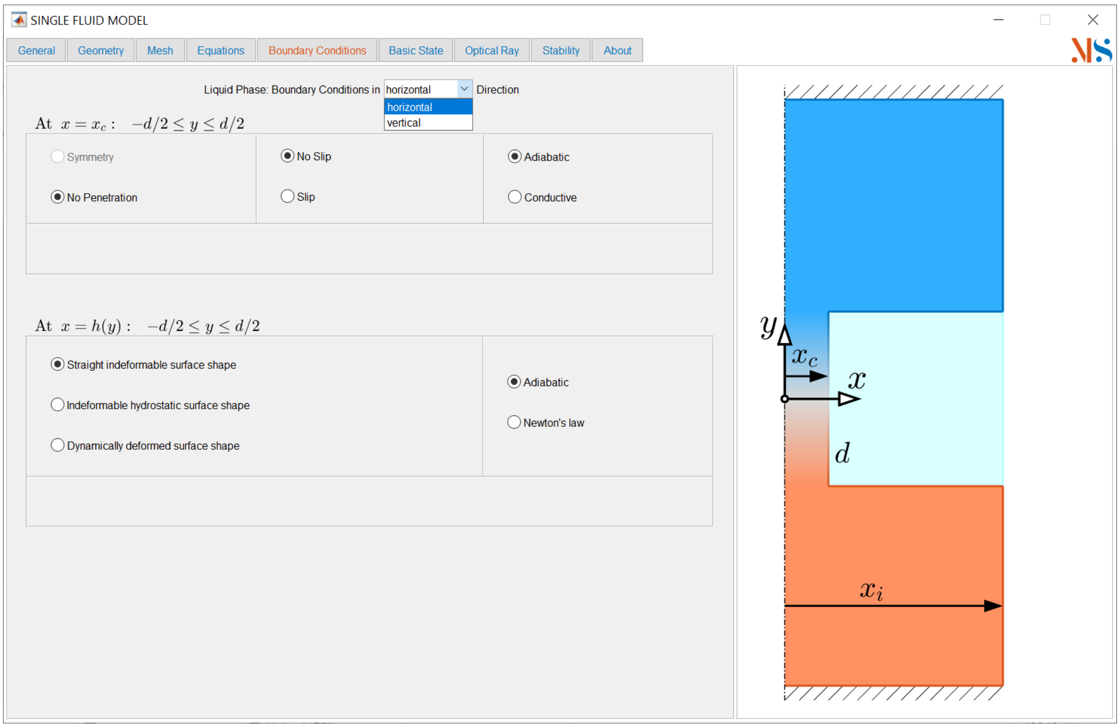
- Leave both, the energy equation and the thermocapillary convection enabled.



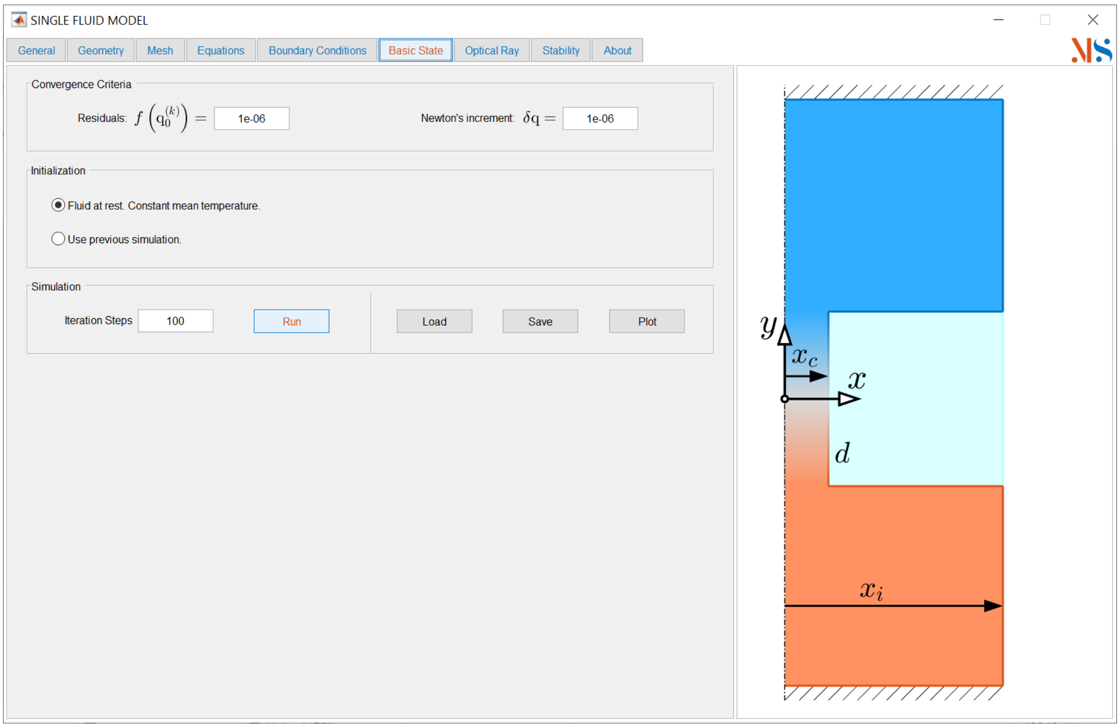
- Set the boundary conditions for the vertical direction. A temperature difference of $|\Delta T| = |T_{d1} - T_{d2}| = 10\,\mathrm{K}$ exactely corresponds to $\mathrm{Re} = 1000$ for our customized fluid.



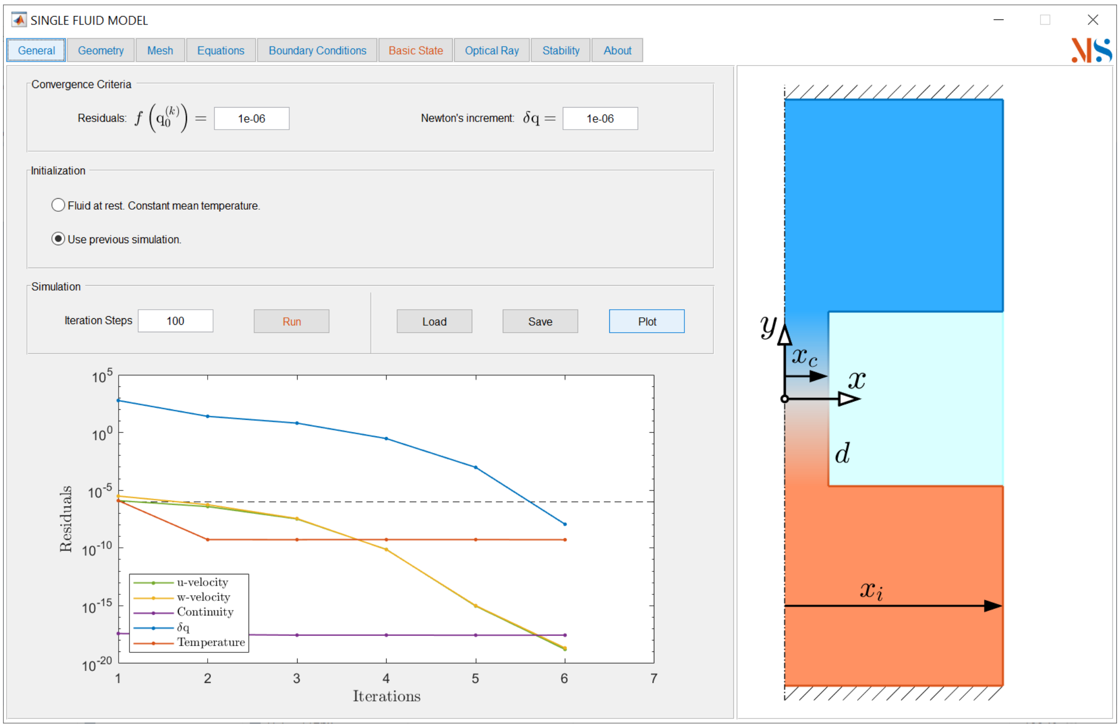
- Set the boundary conditions for the horizontal direction. Prohibit surface deformations by ticking the box `Straight indeformable surface shape`.



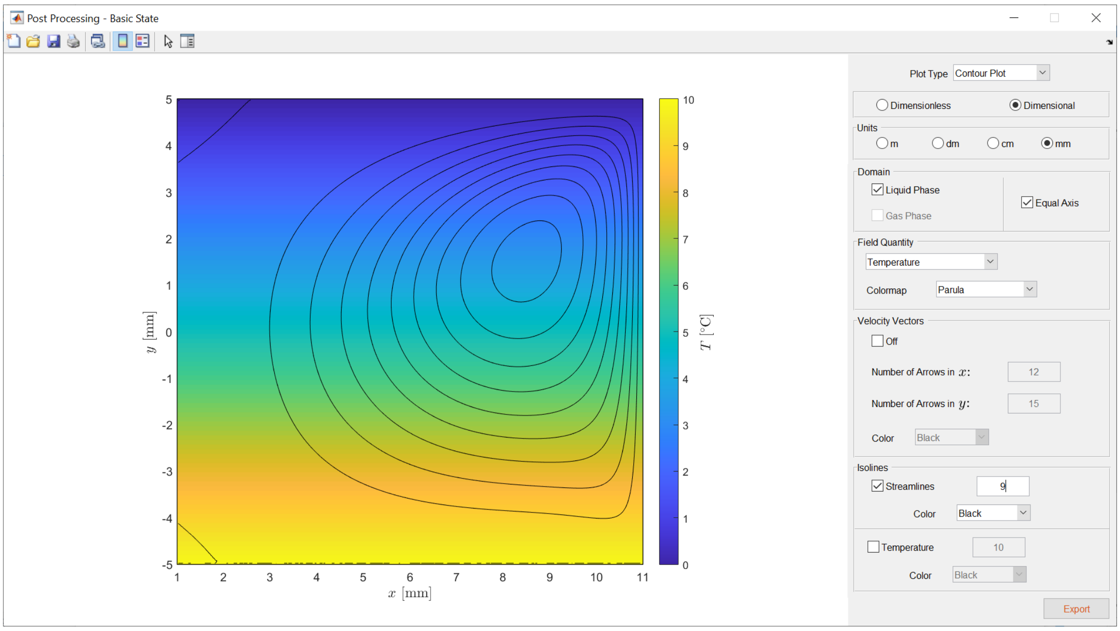
- Set the tolerances of the residuals for the Newton solver and the solution increments.



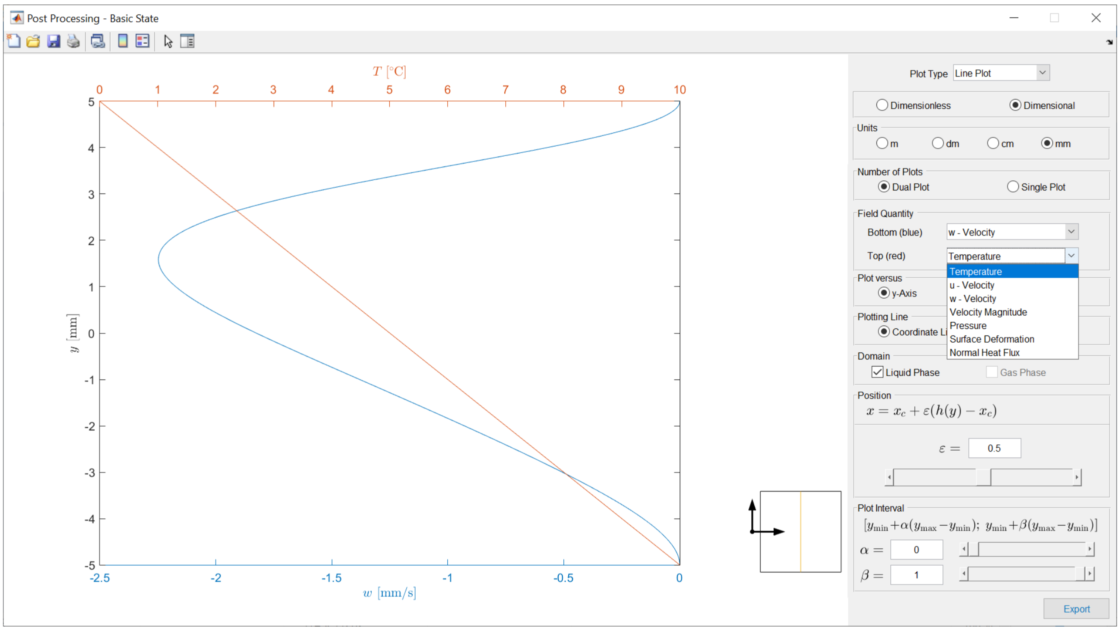
- Solve the basic state by clicking on `Run`.



- Click on `Plot` to visualize the basic state. Choose the desired field quantity and isolines. Check out the different colormaps provided.



- Change the plot type to `Line Plot` and change the position (location at which the data are evaluated, yellow line) to e.g. $\varepsilon = 0.5$ to visualize the temperature and vertical velocity component at the vertical midplane. As expected, the temperature profile is perfectly linear. The velocity $w(y)$ is asymmetric due to the inertia effect present for $\mathrm{Re} = 1000$.



References

He, J., Liu, W. and Huang, Y.-X. (2016), 'Simultaneous determination of glass transition temperatures of several polymers', *PLoS ONE* **11**, 0151454 (12pp).

Joseph, D. D. (1976), *Stability of Fluid motions I*, Vol. 27 of *Springer Tracts in Natural Philosophy*, Springer, Berlin, Heidelberg.

Romanò, F. and Kuhlmann, H. C. (2017), 'Particle–boundary interaction in a shear-driven cavity flow', *Theor. Comput. Fluid Dyn.* **31**, 427–445.

Stojanović, M., Romanò, F. and Kuhlmann, H. C. (n.d.), 'High-Prandtl-number thermocapillary liquid bridges with dynamically deformed interface: Effect of an axial gas flow on the linear stability (unpublished)'.