Case 1: a simple Rankine cycle

Introduction

Cycle-Tempo is a powerful software program designed for thermodynamic modeling and optimization of systems that generate electricity, heat, and refrigeration. These systems often consist of multiple interconnected cycles, each containing various components, which are linked by pipes, forming a complex network of mass and energy flows.

The primary function of Cycle-Tempo is to calculate the key mass and energy flows within such systems. The program offers unmatched flexibility by allowing users to define the system configuration. The types of apparatuses and their interconnections can vary widely between systems, making it crucial for users to customize their setups.

Cycle-Tempo features a comprehensive library of models for apparatuses and pipes, enabling users to create tailored system models. This flexibility sets Cycle-Tempo apart from many other programs, which often restrict or disallow changes to system configurations.

How to start

To get started, we recommend spending five minutes watching the tutorial available on the home page after logging in. Then, follow these steps to set up an energy conversion system in Cycle-Tempo:

1. **Configure Your System**:
   Add components to the canvas and connect them using pipes (and shafts). Refine the layout by rearranging components and pipes and adjusting colors, line thickness, and other visual elements.

2. **Define Working Fluids and Properties**:
   Specify the working fluids (if necessary) and enter their thermodynamic properties on the property pages.

3. **Run the Solver**:
   Once sufficient data has been entered, run the solver to calculate unknown system data.

4. **View Results**:
   Use the result tool ( ) to display results directly on the process diagram (see figure) or access detailed calculations via the **Text Output** tab.

Explanations and tips

1. **Saving a System**:
   Double-click the canvas and select the **Ident** tab in the property page (if not already open). The case name here refers to the name under which the system is saved.

2. **How the Solver Works:**
   The primary goal of the Cycle-Tempo solver is to calculate the mass flow rates in all the pipes of an energy conversion system. This is achieved by constructing a system matrix comprising mass and energy balances. Each component added to the system (e.g., boiler, pump, turbine) contributes a specific number of balances to this matrix. The exact number of balances depends on the type of component and, for some components, on

the input data. To create a solvable system, the matrix must contain as many equations as there are pipes in the system, resulting in N equations with N unknown mass flows.

For a system to be solvable, it must neither be under-specified (too few equations) nor over-specified (too many equations). Consequently, careful selection of components and their input data is necessary to maintain a balanced and solvable system.

Examples:

- Specifying a mass flow rate (Φ_m) in a sink/source with one inlet (a sink) or one outlet pipe (a source) adds a mass balance to the system matrix.
- Specifying the power output (W) of a turbine adds an energy balance.
- For certain components, such as the heat exchanger, condenser, and the combustor, the energy balance can be used either to calculate a temperature or to calculate a mass flow rate. If the energy balance is used for the latter, it is included in the system matrix.
- By double-clicking on the canvas and selecting the **Thermo** tab on the property page (if not already open), you can specify a production function, which consequently adds an energy balance.

For further details, refer to the PDF manuals in the **Knowledge Center** in Cycle-Tempo Online. The PDF manuals include:

- **Technical Notes**: Explains the theoretical background.
- **Reference Guide**: Provides input data details, including component connections and their contributions to the system matrix.

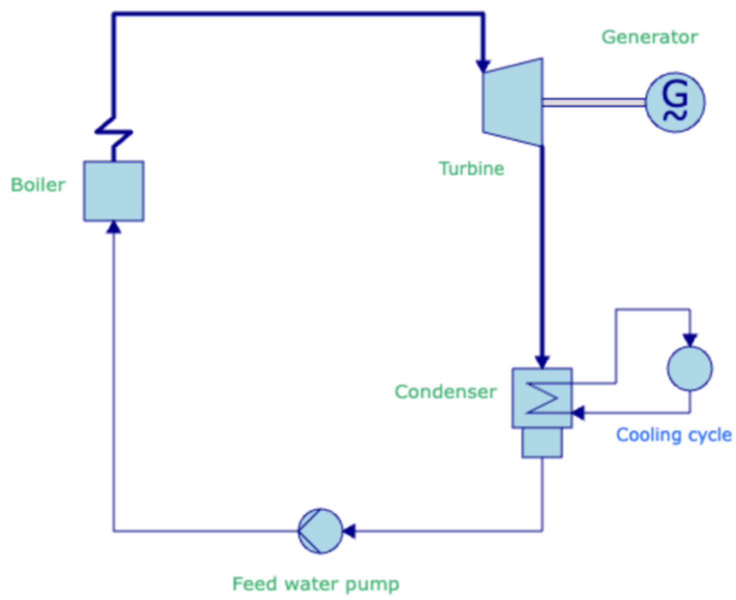
3. **Working Fluids:**
Working fluids are specified through the pipes. For each independent cycle, at least one working fluid must be defined. However, it is only necessary to specify the fluid for one pipe if the composition remains unchanged throughout the cycle.

    - For a close cycle it does not matter through which pipe you enter the working fluid
    - For an open cycle, you need to select the pipe most stream upwards in the system (e.g. the one connected the sink/source with only an outgoing pipe).

When a pipe is represented with the symbol for liquid or vapor, the default working fluid is water/steam.

The simple Rankine cycle case

In the boiler superheated steam is generated. This steam expands in a steam turbine, which drives a generator. The steam in then condensed into water in the condenser. The cooling water circuit is modelled just by a sink/source (you are challenged to add a cooling water pump). The condensate from the condenser is pumped by the feed water pump to the boiler. In the figure below you find a layout of this simple Rankine cycle:



Starting points for the calculation:

Working fluids:
- main cycle: water/steam
- cooling cycle: water

Steam turbine
- inlet pressure 40 bar
- inlet temperature 400 °C
- isentropic efficiency 85 %
- power 1 MW

Condenser
- main flow
  - outlet pressure 0.05 bar
  - outlet condition fluid saturated liquid
  - pressure loss 0 bar

cooling flow

ΔT cooling water	7 ºC
pressure loss	0 bar

Boiler feed pump

isentropic efficiency	70 %

Boiler

pressure loss	1 bar

Generator

Efficiency	95 %

After successfully entering the data and performing a calculation, you can display the results directly within the process scheme, as shown in the figure below. Additionally, all calculated results can be reviewed in detail under the **Text Output** tab.