

The background features a semi-transparent orange overlay on a scene with a car and an airplane. The car is in the foreground, and the airplane is in the background. Both are overlaid with a grid of simulation data points and numerical values, suggesting a finite element analysis or simulation process.

# SIMULATION AND POSTPROCESSING

Lecture 1.2

*Modelon*

# OVERVIEW

✓ Setting up an experiment model

✓ Running multiple experiments

✓ Dynamic Solvers Settings

✓ Results

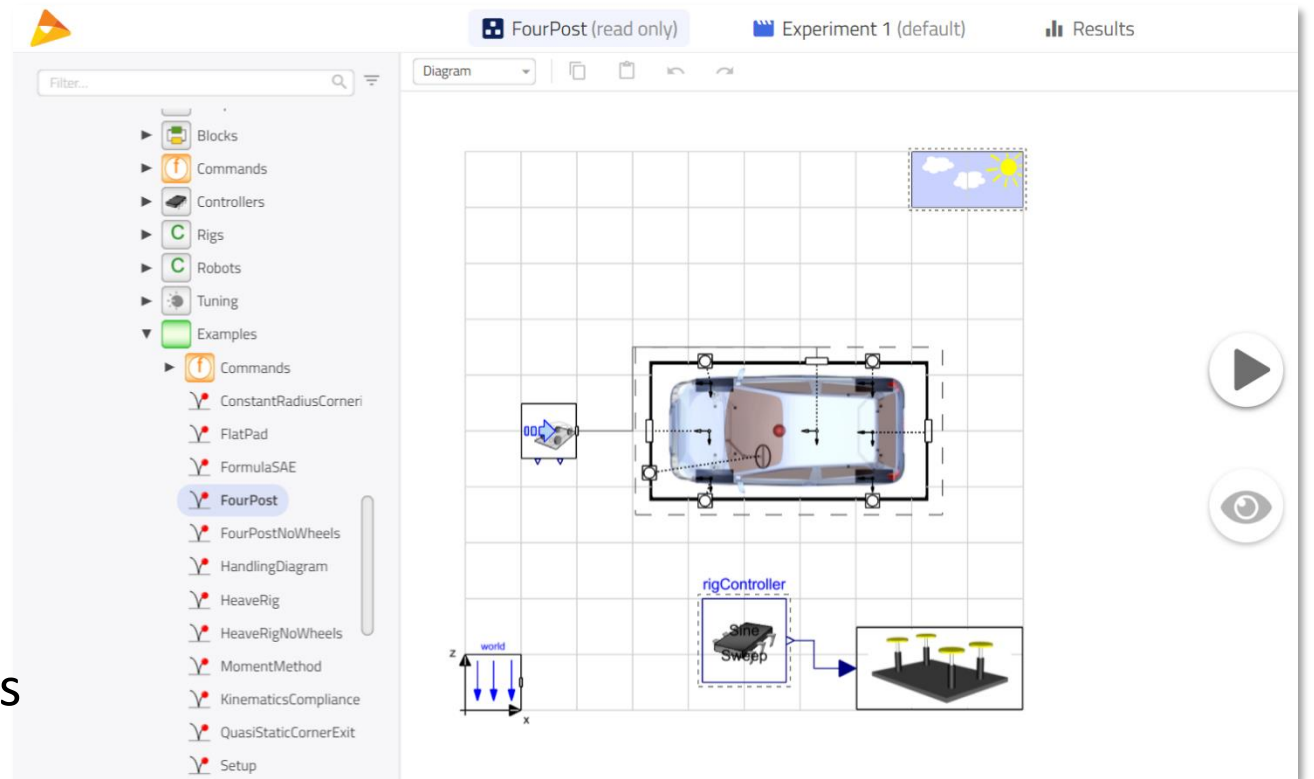
✓ Stickies and views



# SETTING UP AN EXPERIMENT MODEL

# WHAT IS AN EXPERIMENT MODEL

- A system model with a purpose:
  - Specific data set for parameters
  - Boundary conditions applied
  - Initial conditions applied
  - Can be executed
  - Intention to retrieve results
- Good practice to store separately
- Indicate for users that they are executables



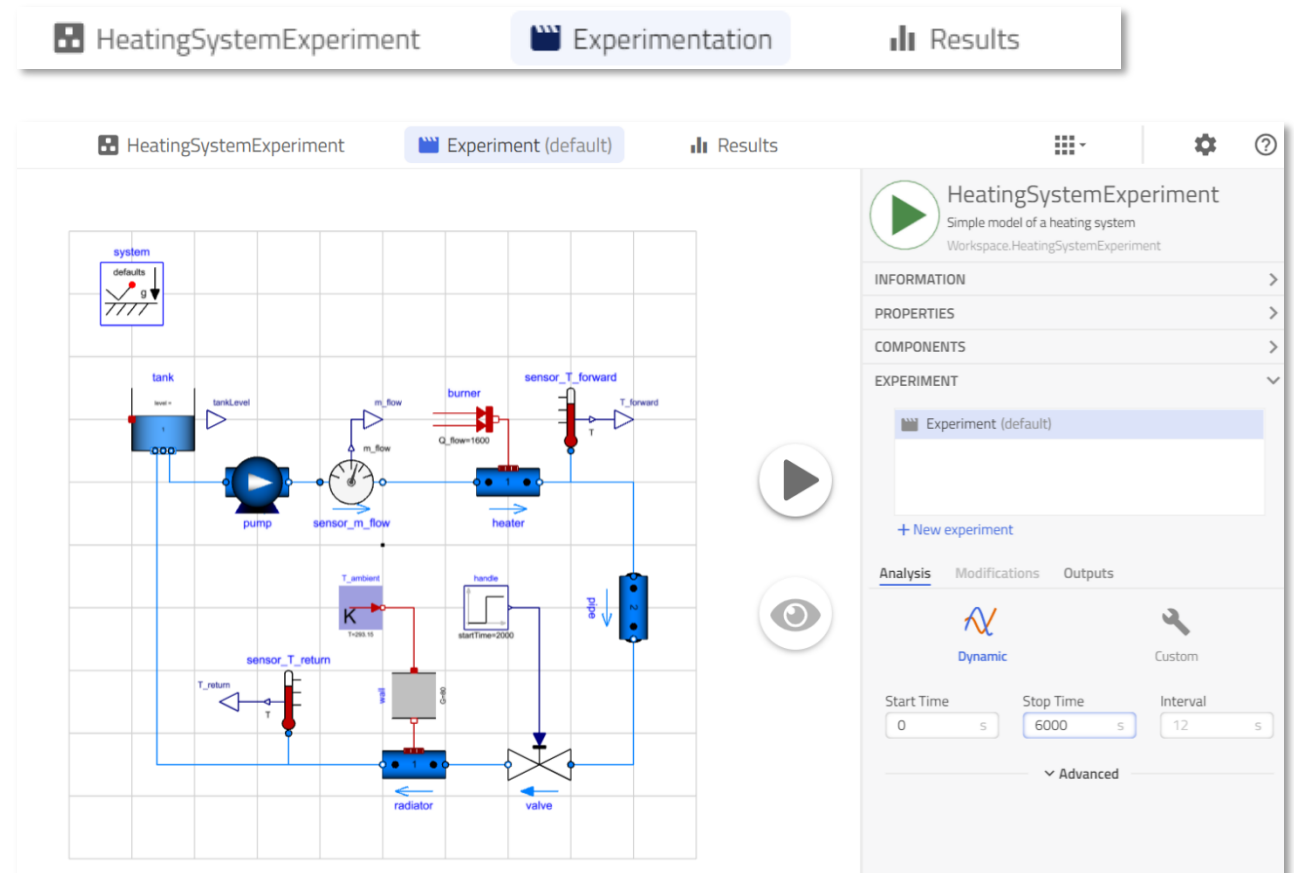


# RUNNING MULTIPLE EXPERIMENTS

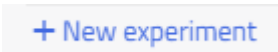
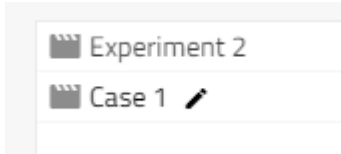
# EXPERIMENTATION MODE

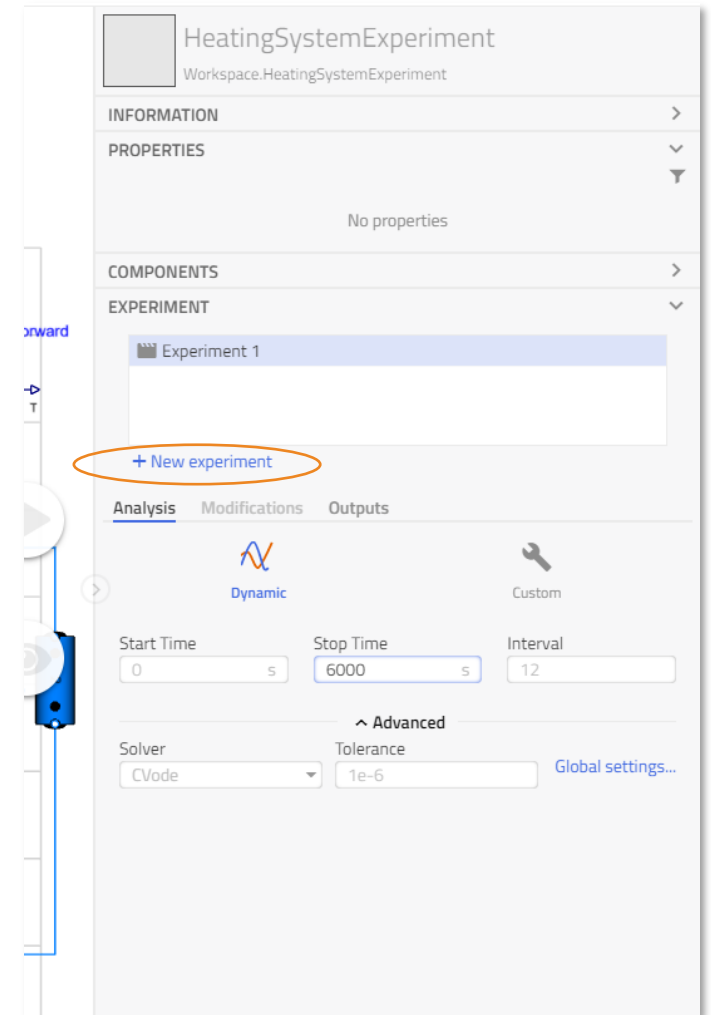
- Impact makes it easy to run several experiments on the same system model
- Each experiment can have separate
  - Data modifications
  - Analysis type
  - Solver settings
  - Compiler settings

When clicking the run button you implicitly run an experiment



# CREATE NEW EXPERIMENT

- Create several new experiment with 
- Experiments can be renamed 
- When a specific experiment is chosen, all changes to the model in that mode, will be saved as modification to the model and only applied to that experiment.



# CHANGES IN EXPERIMENTS

- In the example, Case 1 is active, and variable `p_a_nominal` has changed its value
- All parameter changes in a specific experiment can be viewed under "Modifications" tab

The screenshot shows the Modelon software interface for a heating system experiment. The main workspace displays a schematic diagram of the system, including a tank, a pump, a burner, a heater, a radiator, and various sensors. The right-hand panel shows the properties of the selected 'pump' component, which is a centrifugal pump with an ideally controlled mass flow rate. The 'Modifications' tab is active, showing a list of changes to the pump's parameters.

Parameter	Value	Unit
<code>p_a_nominal</code>	210000	Pa
<code>p_b_nominal</code>	130000	Pa
<code>m_flow_nominal</code>	0.01	kg/s
<code>control_m_flow</code>	<input type="checkbox"/>	»
<code>use_m_flow_set</code>	<input type="checkbox"/>	»
<code>use_p_set</code>	<input type="checkbox"/>	»

The screenshot shows the 'EXPERIMENT' panel in Modelon. It displays a list of experiments, with 'Case 1' selected. Below the list, there is a '+ New experiment' button. The 'Modifications (1)' tab is active, showing a table of modifications for the selected experiment.

Modification	Value
<code>pump.p_a_nominal</code>	210000



# WORKFLOW

From one system model, you can define several experiments to generate results

System model -> Experiment -> Results





# DYNAMIC SOLVER SETTINGS

# SOLVING A DYNAMIC PROBLEM

- The resulting mathematical problem formulation is an ODE (or DAE).

$$\dot{x} = f(x, u)$$

- Modelon Impact has a set of ODE solvers to chose from.

The screenshot displays the 'Analysis' tab of the Modelon Impact software. The interface is divided into three main sections: 'Analysis', 'Modifications (1)', and 'Outputs'. Under the 'Analysis' section, there is a 'Dynamic' icon and a 'Custom' icon. The 'Start Time' is set to 0, the 'Stop Time' is set to 6000, and the 'Interval' is set to 12. Below these settings, there is an 'Advanced' section with a dropdown menu for 'Solver' (currently set to 'CVode') and a 'Tolerance' input field set to '1e-6'. A 'Global settings...' link is also visible.

# SELECTING SOLVERS

- Variable step size
  - Most integration algorithms available have a *variable step size algorithm*.
  - The integration step size is chosen in such a way, that the error is smaller than the desired maximum error, defined via the set tolerances.
  - This implies, that usually smaller step sizes are used, if smaller tolerances are defined.
- One-step algorithms versus multi-step algorithms
  - One-step algorithms (like Radau) are basically designed such that they start fresh on every step
    - The cost of restarting them after an event is substantially reduced
  - Multi-step algorithms such as CVODE base the next step upon previous steps
    - Expensive to restart the simulation after an event

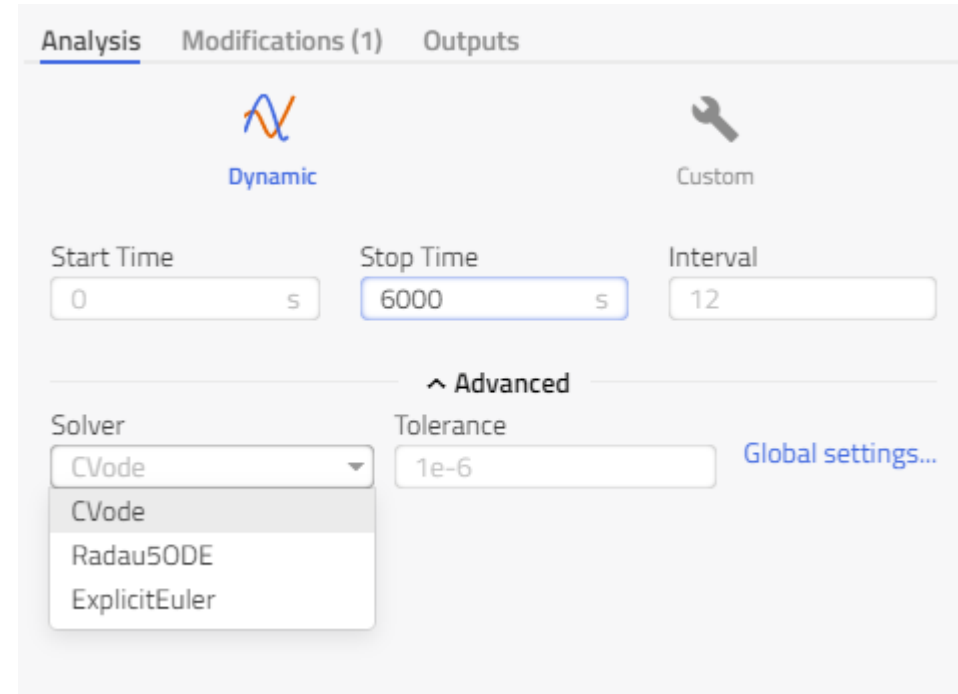
# SELECTING SOLVERS

- **CVODE** is a solver for stiff and non-stiff ordinary differential equation (ODE) systems (initial value problem) given in explicit form  $y' = f(t,y)$ . The methods used in CVODE are **variable-order, variable-step, multistep** methods. For nonstiff problems, CVODE includes the Adams-Moulton formulas, with the order varying between 1 and 12. For stiff problems, CVODE includes the Backward Differentiation Formulas (BDFs) in so-called fixed-leading coefficient form, with order varying between 1 and 5.
- **Radau5ODE**: Radau IIA fifth-order three-stages with step-size control and continuous output. Based on the FORTRAN code RADAU5 by E.Hairer and G.Wanner.
- **ExplicitEuler**: This solver solves an explicit ordinary differential equation using the explicit Euler method.

The screenshot displays the 'Analysis' tab in the Modelon software. At the top, there are three icons: 'Dynamic' (a blue graph icon), 'Steady State' (a target icon), and 'Custom' (a wrench icon). Below these are three input fields: 'Start Time' with a value of 0, 'Stop Time' with a value of 1, and 'Interval' with a value of 0.002. A section titled 'Advanced' is expanded, showing a 'Solver' dropdown menu with 'CVode' selected, and a 'Tolerance' input field with a value of 1e-6. A 'Global settings...' link is visible on the right side of the 'Advanced' section.

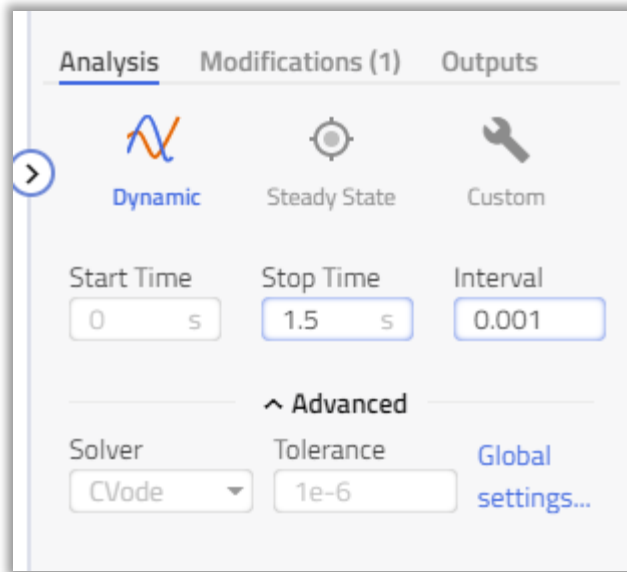
# SELECTING SOLVERS

- Tolerance: affects the accuracy of the solution
- Interval: affects the resolution of the stored solution

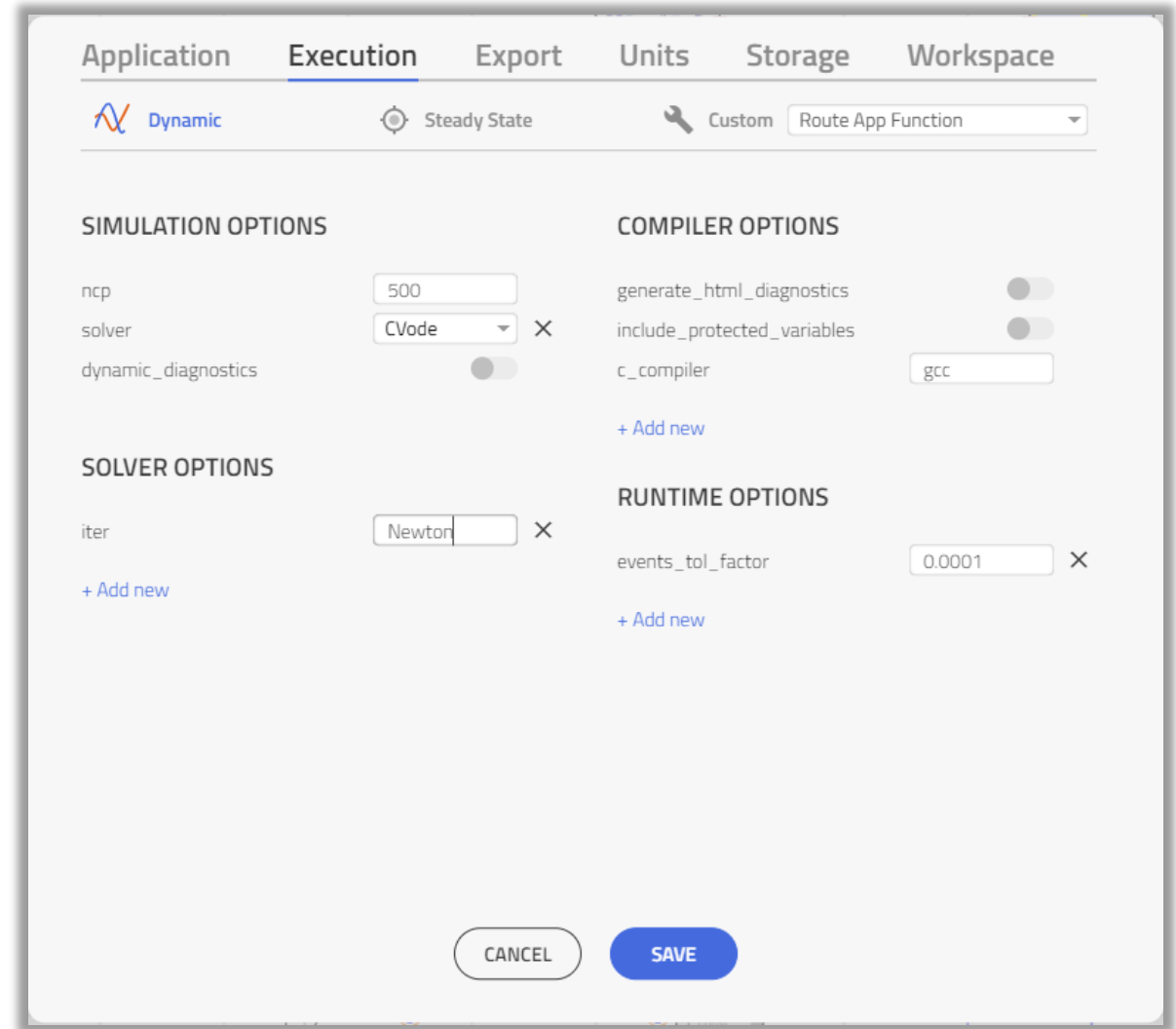


# SETTINGS

- Found under  or “Global settings...”



- Detailed information found in Help Center:
  - OPTIMICA Compiler Toolkit User’s Guide



# SELECTING SOLVERS

- Should not rely on just one integration algorithm for simulation experiments.
  - Instead, some selected results should be checked using different integration algorithms to find out which works best.
    - CPU time
    - Stability
    - Accuracy



# SIMULATION BUTTON VISUAL FEEDBACK

- Progression:

Compiling...



Simulating  
(7 / 25)...



Simulating...



- Compiled model available:

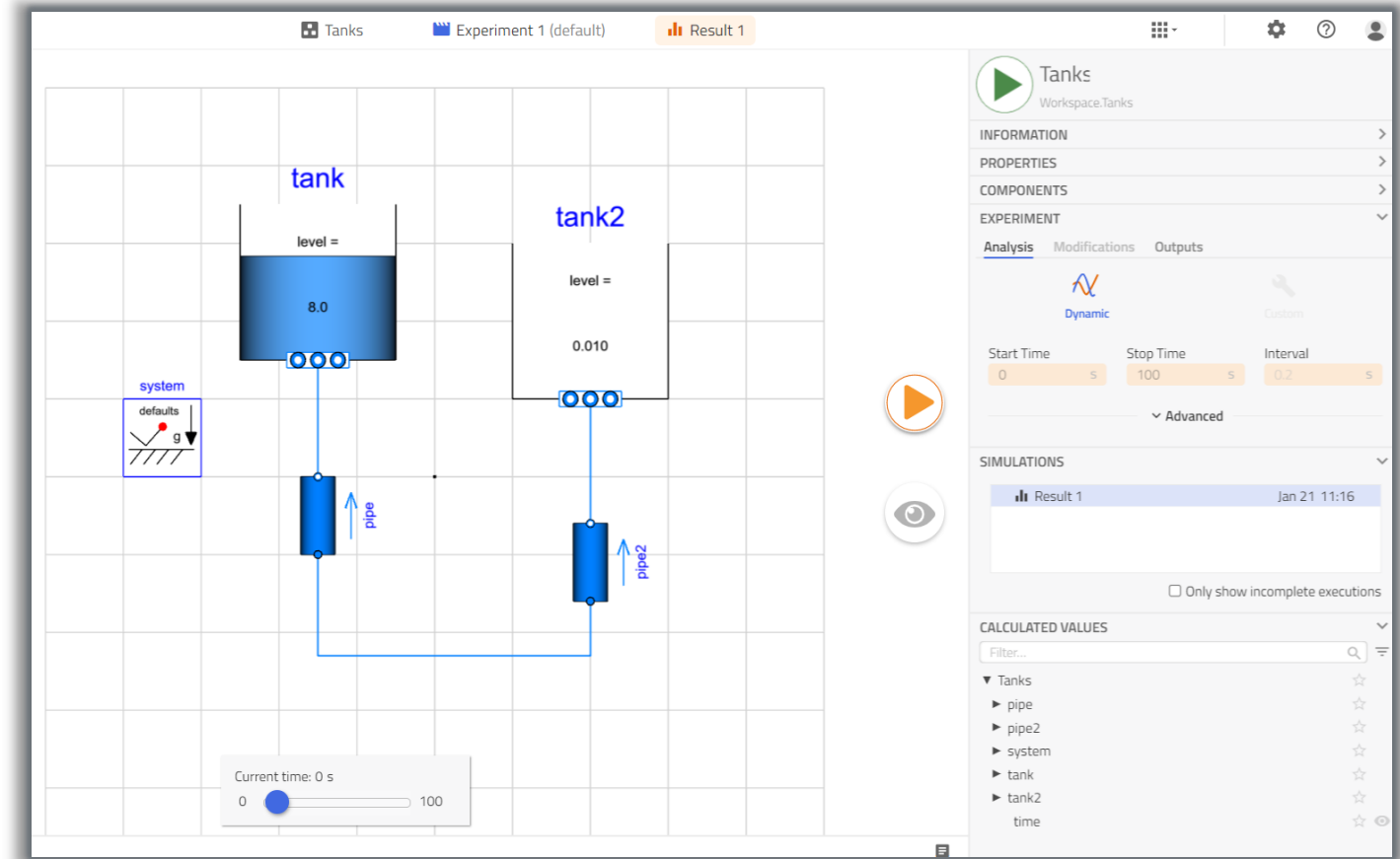
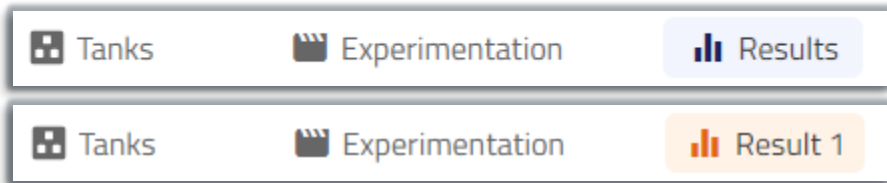


- Result available for viewing:

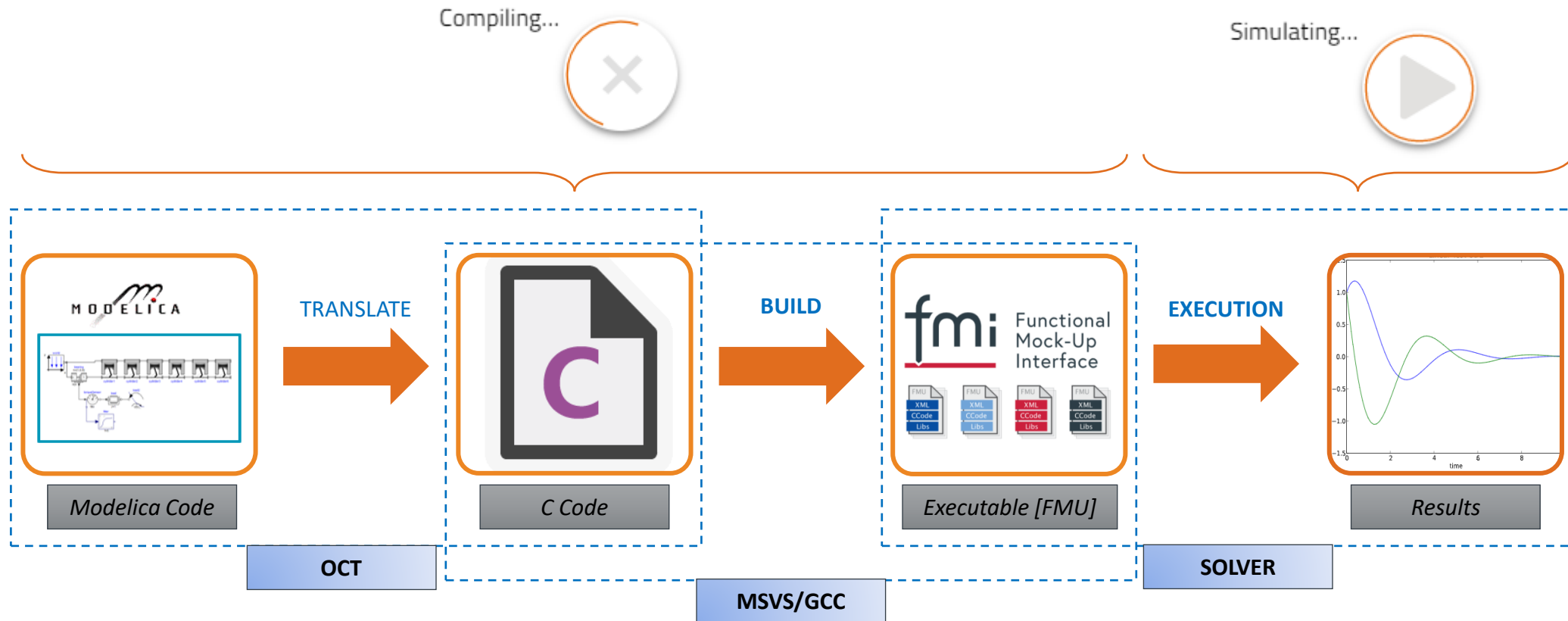


# SIMULATION

- When simulation is finished, a result becomes available for analysis in the **Simulations** list.
- By default, the latest result is selected
- Variables organized according to components are listed in **Calculated Values**
- Once simulation is completed, the mode is automatically switched to Results and the name is updated based on the selected result



# WHAT HAPPENS BEHIND THE SCENES



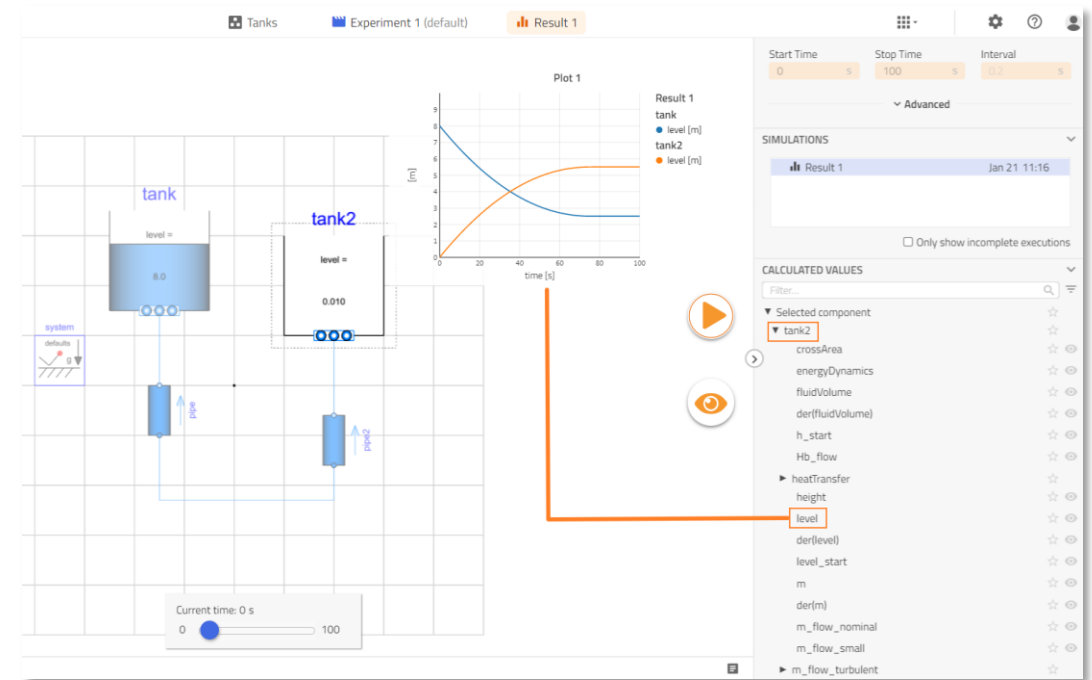
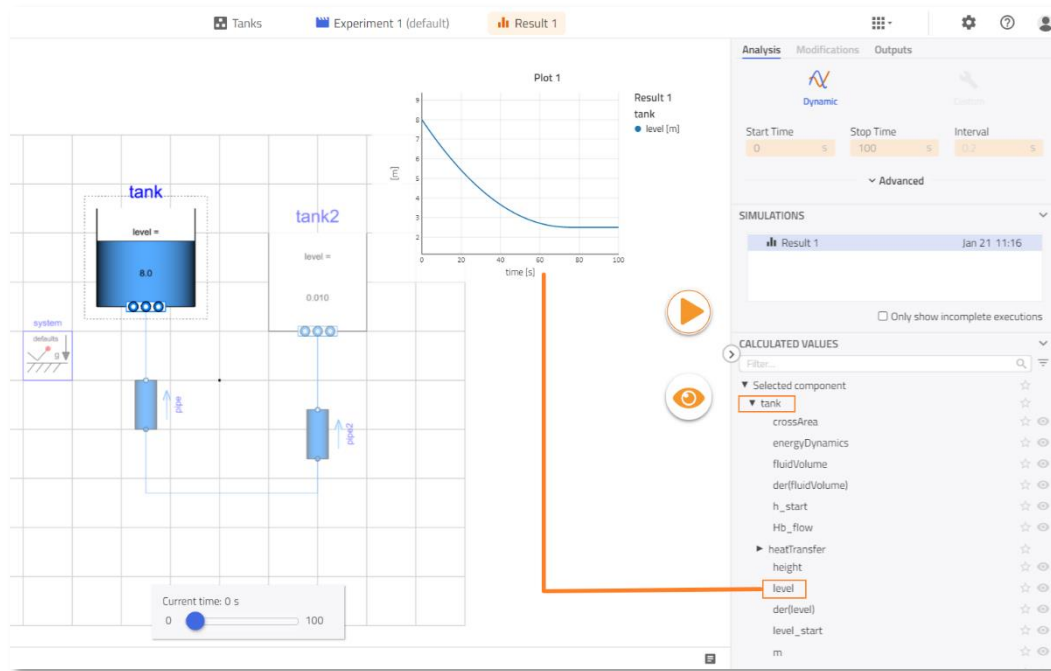


# RESULTS

# INSPECTING RESULTS: PLOTS

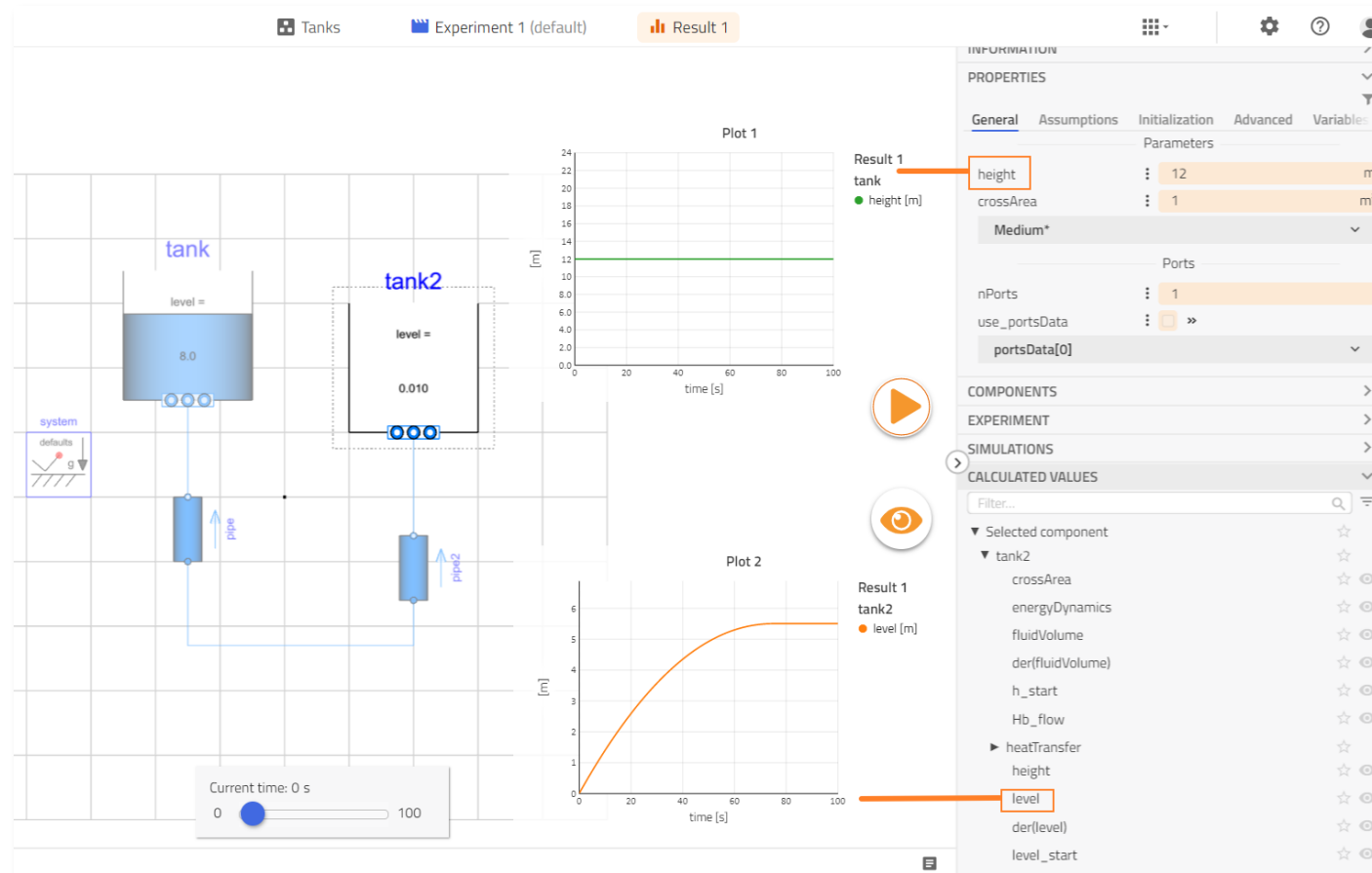
- To plot a variable, simply drag a variable on to the canvas from the variable list

- Multiple variables can be dropped on the same plot for comparison



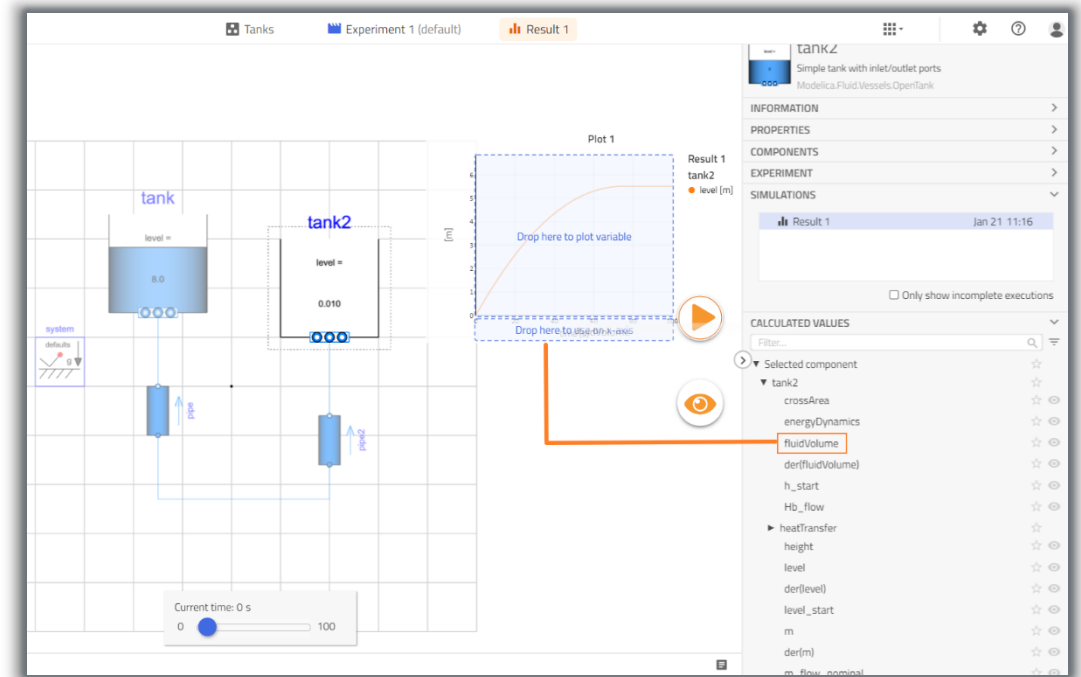
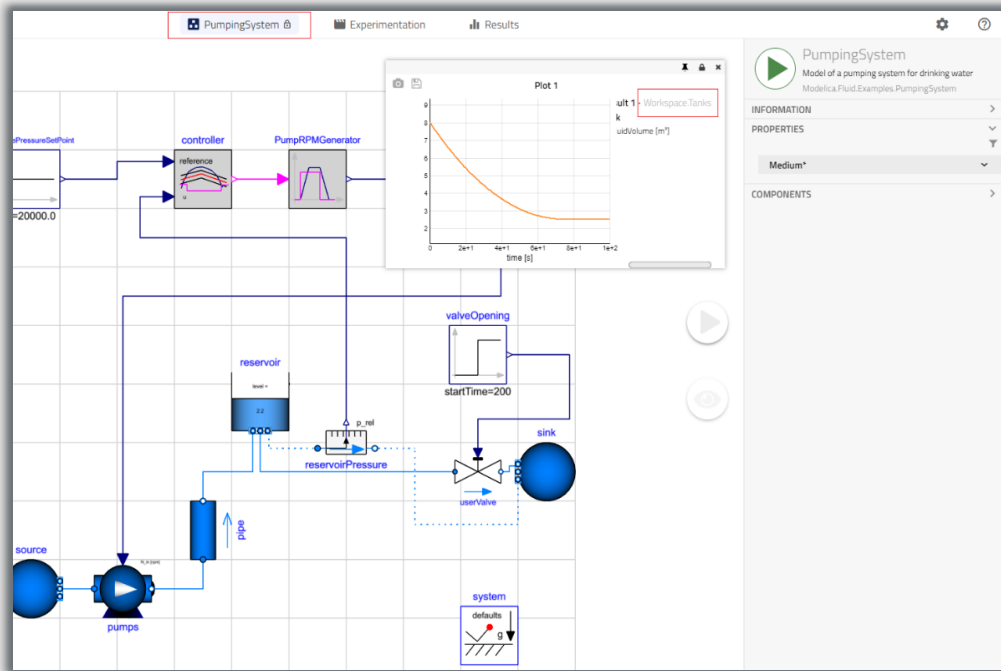
# INSPECTING RESULTS: PLOTS

- Parameters and variables can also be dragged to be plotted on the canvas



# INSPECTING RESULTS: PINNED AND X-Y PLOTS

- Plots can be pinned such that they remain on the canvas even if the model is switched
  - Useful for comparing results between models
- Independent variable can be changed in order to generate X vs. Y or parametric plots

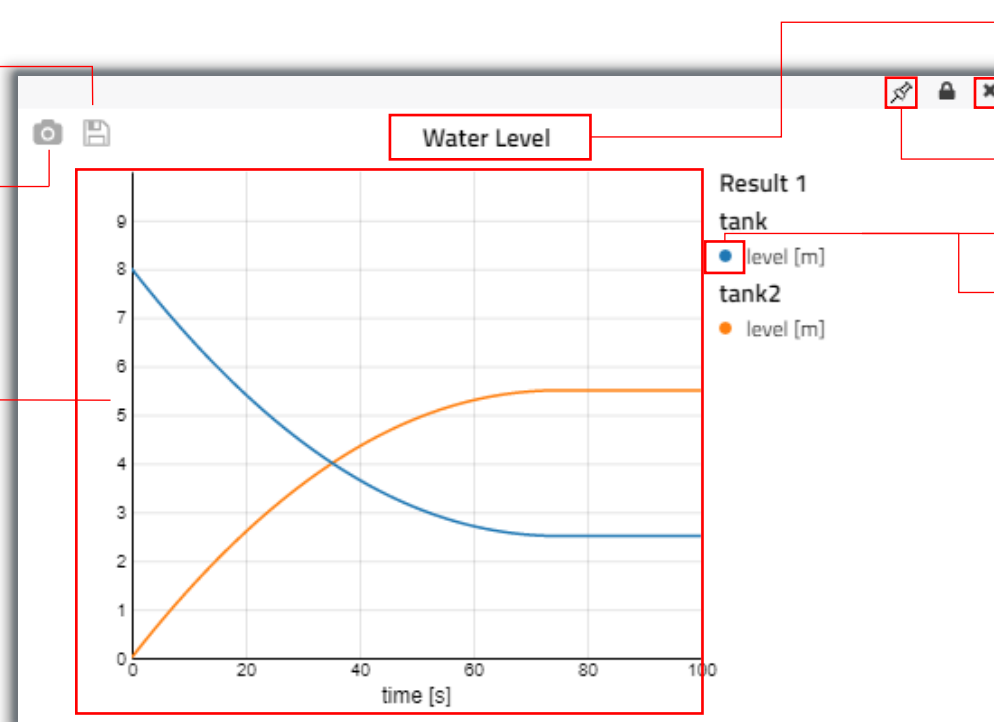


# INSPECTING RESULTS: PLOTS - WINDOW FUNCTIONALITY

Download plot

- CSV
- PNG

Zooming capabilities



Rename plot

Remove plot

Pin plot

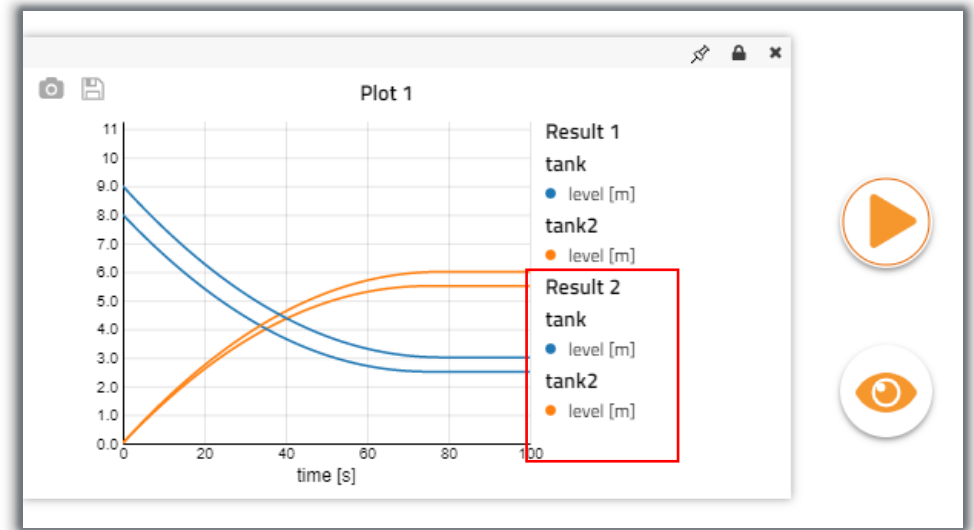
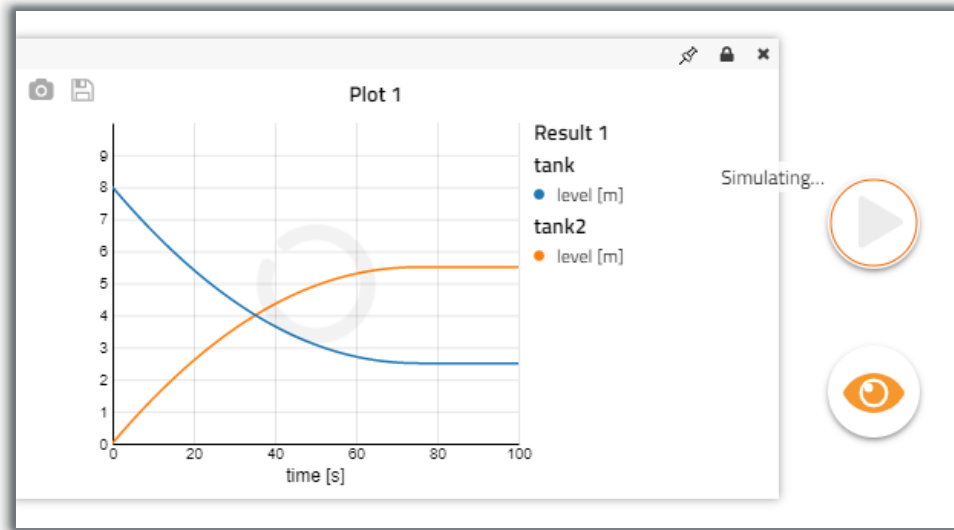
Edit color

Remove trajectory



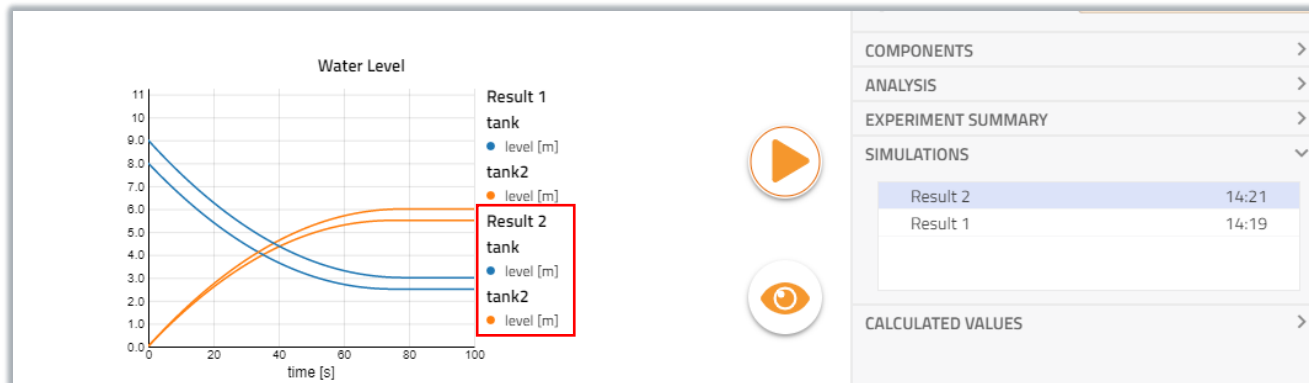
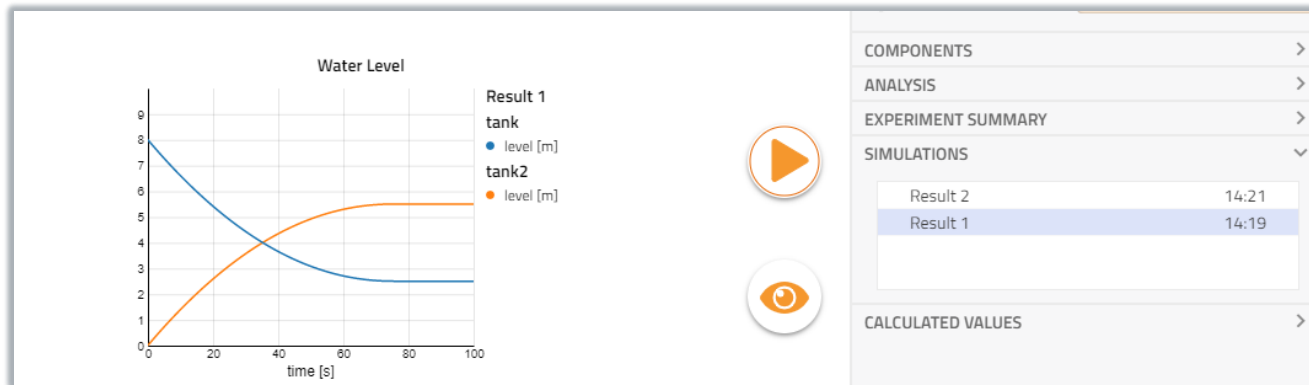
# INSPECTING RESULTS: PLOTS – COMPARING RESULTS

- When running a new simulation with the plot window open, new results will automatically be displayed in the plot



# INSPECTING RESULTS: PLOTS – COMPARING RESULTS

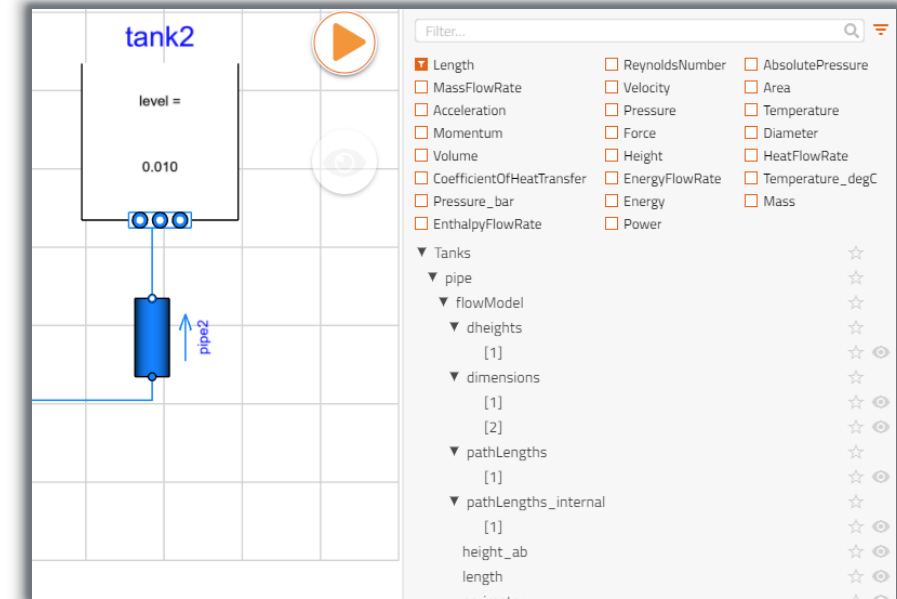
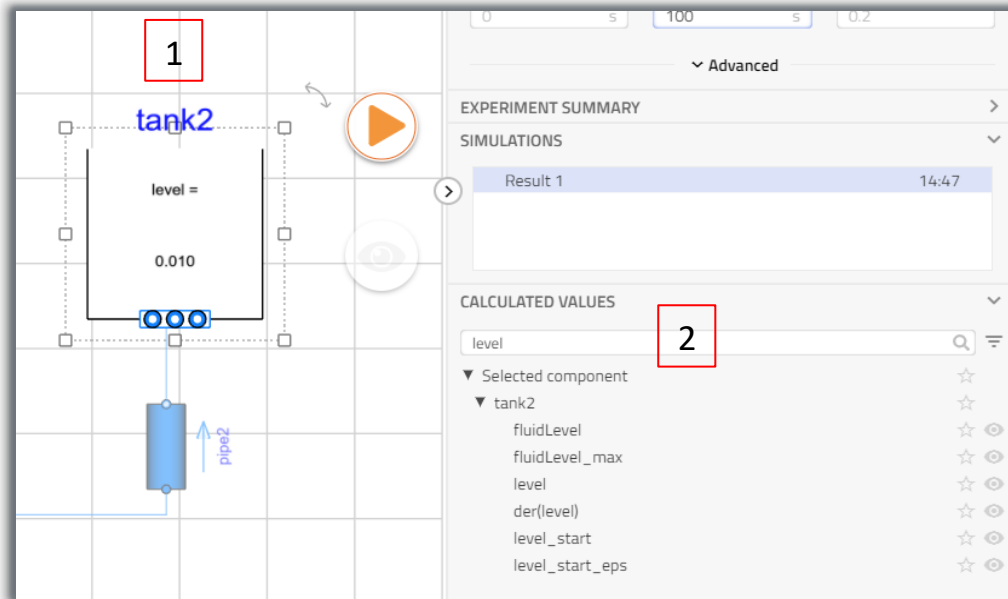
- Compare results by selecting another result while having a plot window open



# INSPECTING RESULTS: FILTERING

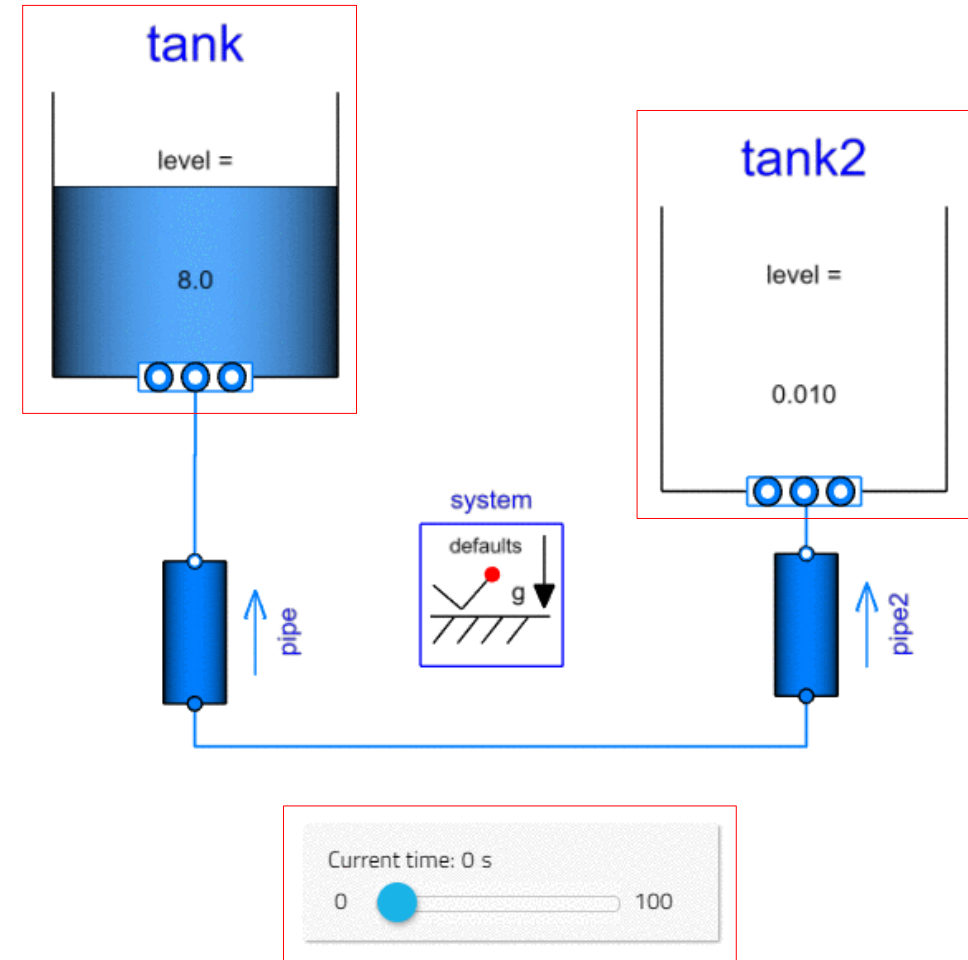
- Find variables by filtering. Three options exist:
  1. Selecting component in canvas
  2. Text filtering
  3. Type filtering
- Filters are additive, only giving results matching all conditions.

3 Type filtering



# INSPECTING RESULTS: MODEL ANIMATION


- Certain models support visualization (ex. *OpenTank* from MSL)
- Model changes appearance when sliding the time-slider

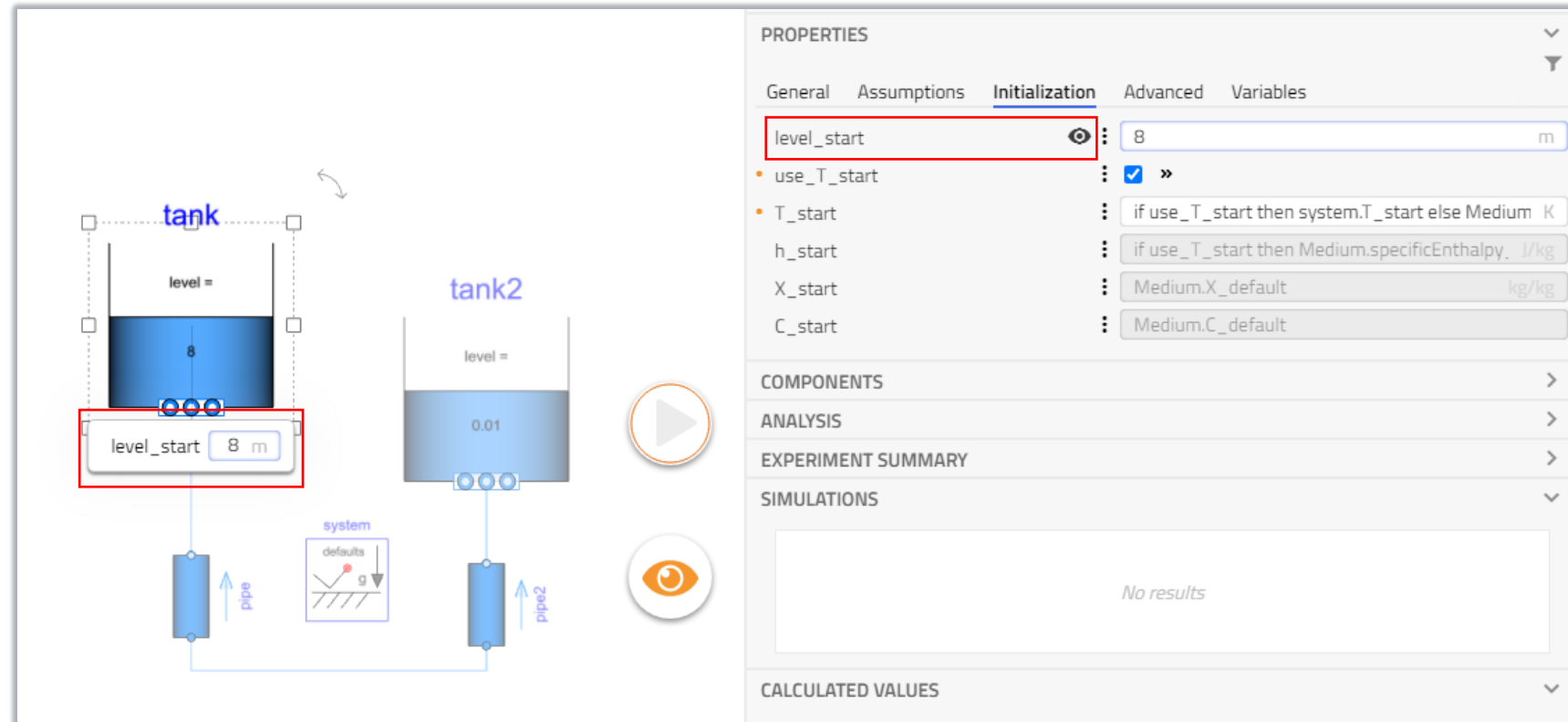




# STICKIES AND VIEWS

# STICKIES

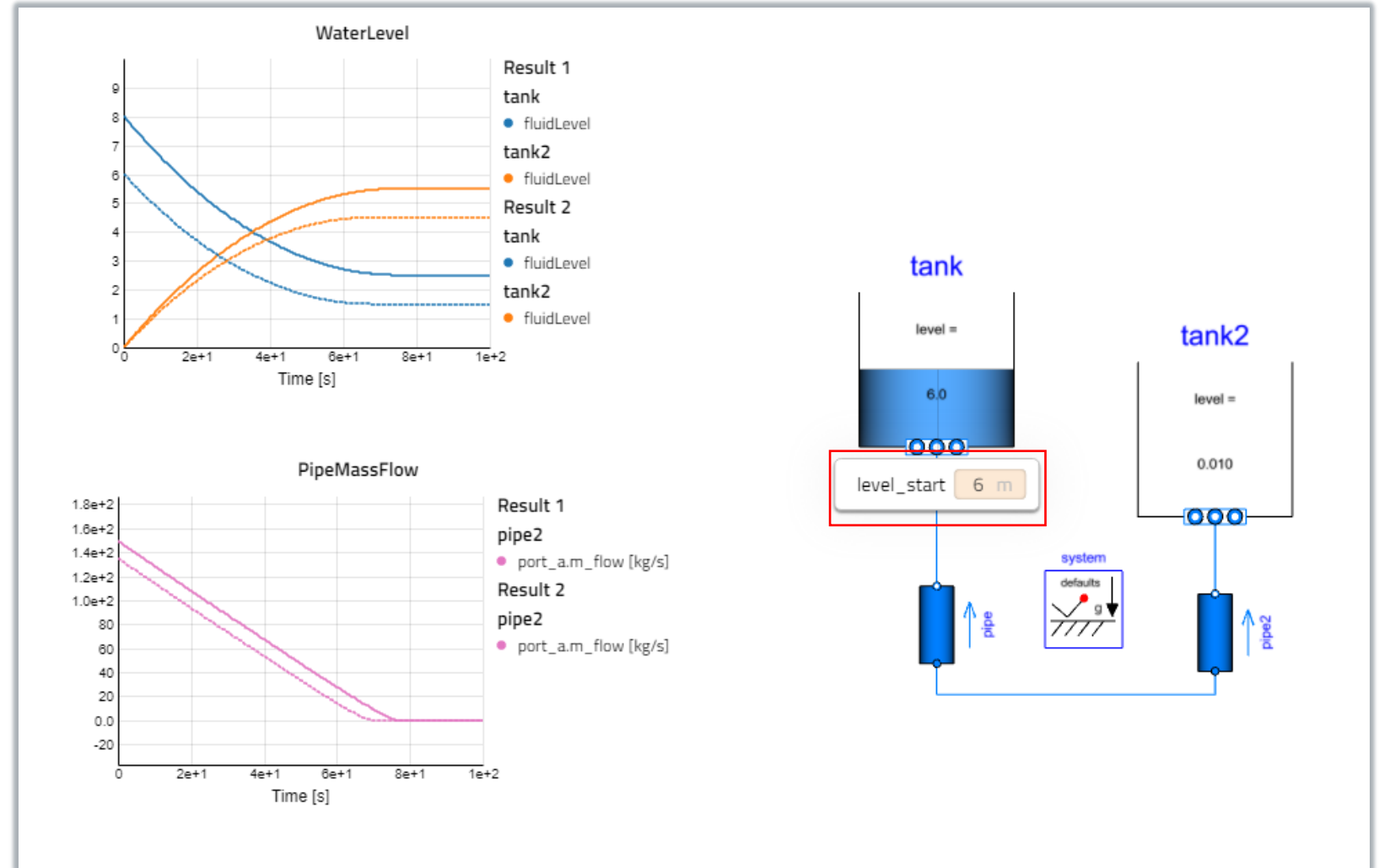
- Stickies are small widgets that can be added to the model canvas for interaction with the model
- Stickies are added with the -button in the control panel
- Modelon Impact-unique feature



The screenshot displays a Modelon Impact interface. On the left, a process flow diagram shows two tanks, 'tank' and 'tank2', connected by pipes. The 'tank' has a level of 8, and 'tank2' has a level of 0.01. A 'stickie' widget is attached to the 'tank' component, showing 'level\_start' set to 8 m. The stickie is highlighted with a red box. To the right, the 'PROPERTIES' panel is open, showing the 'Initialization' tab. The 'level\_start' property is highlighted with a red box and has an eye icon next to it, indicating it is visible. Other properties listed include 'use\_T\_start', 'T\_start', 'h\_start', 'X\_start', and 'C\_start'. The 'COMONENTS', 'ANALYSIS', 'EXPERIMENT SUMMARY', and 'SIMULATIONS' sections are also visible, with 'No results' shown in the simulations section.

# STICKIES

- Can be used to provide input to the model
- Equivalent to parameter input in control panel



# STICKIES

- Can be used to display simulation results
- Moving the slider will update the stickies
- Both parameters and variables can be displayed in the stickies

The screenshot displays the Modelon software interface for a simulation titled "Tanks". The main workspace shows a schematic diagram of two tanks, "tank" and "tank2", connected by pipes. The "tank" has a fluid level of 6.0 m and a mass of 5973.52 kg. The "tank2" has a fluid level of 0.010 m and a mass of 9.95586 kg. The diagram includes a "system" block with a gravity vector "g" pointing downwards. Below the schematic are two graphs: "WaterLevel" and "PipeMassFlow". The "WaterLevel" graph shows the fluid level in meters over time for two results, Result 1 and Result 2. The "PipeMassFlow" graph shows the mass flow rate in kg/s over time for two results, Result 1 and Result 2. A time slider at the bottom indicates the current time is 0 s. The right-hand sidebar contains several panels: "INFORMATION", "PROPERTIES", "COMPONENTS", "ANALYSIS", "EXPERIMENT SUMMARY", "SIMULATIONS", and "CALCULATED VALUES". The "ANALYSIS" panel shows the simulation is set to "Dynamic" mode with a stop time of 100 s and an interval of 0.2 s. The "SIMULATIONS" panel lists two results: Result 2 (14:10) and Result 1 (14:06). The "CALCULATED VALUES" panel shows a search bar and a tree view of the simulation components, with "tank2" selected. The "stickies" (red boxes) are located on the schematic diagram, showing parameters like "level\_start" and "m" for "tank" and "tank2".



# VIEWS

- The views are used to save a set of plots and stickies for a model
- This includes what variables are plotted/shown in stickies as well as the positioning of plots and stickies
- The views are saved in the context of the active workspace for the workspace package and global libraries.

The screenshot displays the Modelon software interface for a simulation titled "Tanks". The main workspace shows a schematic of two tanks, "tank" and "tank2", connected by pipes "pipe" and "pipe2". The "tank" has a fluid level of 6.0 m and a mass of 5973.52 kg. The "tank2" has a fluid level of 0.010 m and a mass of 9.95586 kg. A "system" block is also visible, showing a gravity vector "g".

Two plots are shown at the bottom left:


- WaterLevel**: A line graph showing the fluid level in meters over time (0 to 1000 seconds). The "tank" level (blue line) starts at 6.0 m and decreases to approximately 1.5 m. The "tank2" level (orange line) starts at 0.010 m and increases to approximately 4.5 m.
- PipeMassFlow**: A line graph showing the mass flow rate in kg/s over time (0 to 1000 seconds). The flow rate (pink line) starts at approximately 1.4e+2 kg/s and decreases to 0.0 kg/s.

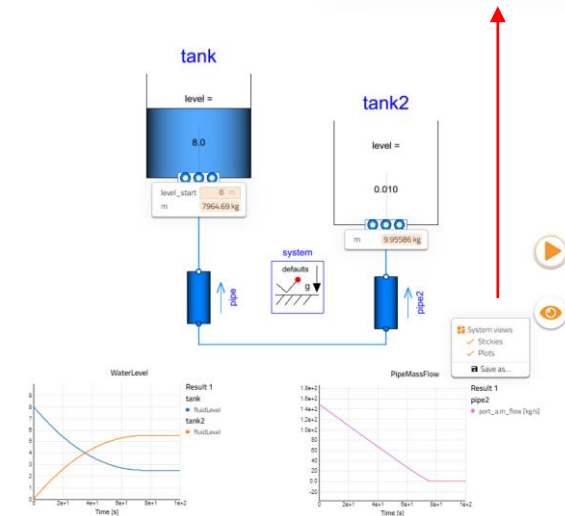
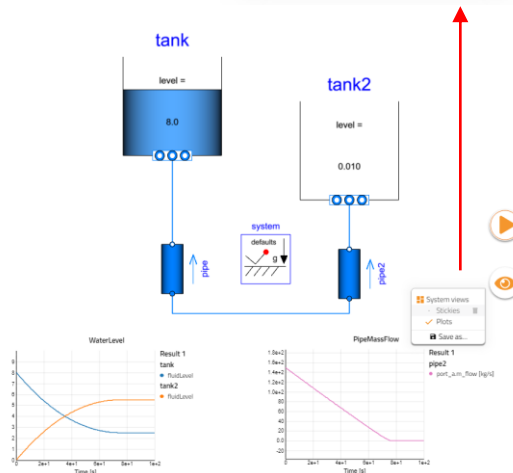
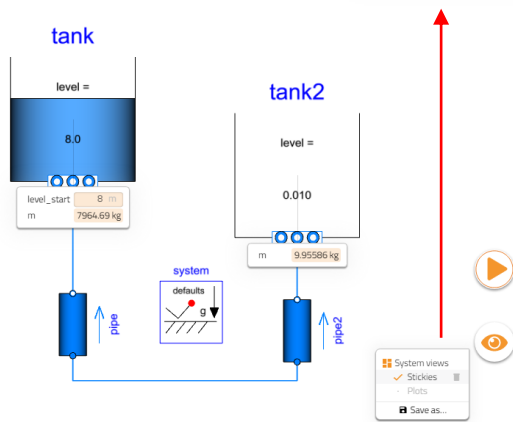
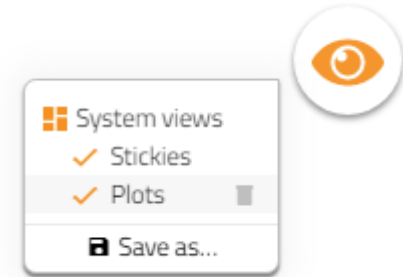
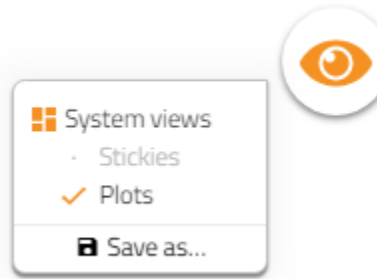
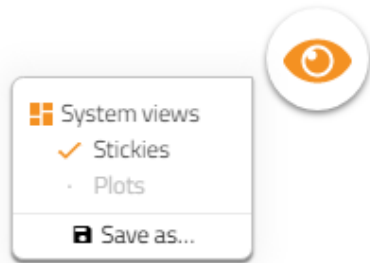
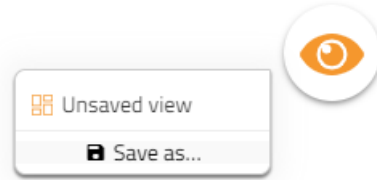
The right-hand sidebar contains several panels:

- INFORMATION**: Workspace: Tanks
- PROPERTIES**: No properties
- COMPONENTS**: Dynamic, Custom
- ANALYSIS**: Start Time: 0 s, Stop Time: 100 s, Interval: 0.2
- EXPERIMENT SUMMARY**: Result 2 (14:10), Result 1 (14:06)
- SIMULATIONS**: (Empty table)
- CALCULATED VALUES**: m

A red box highlights a "System views" menu with options for "Demo" and "Save as...".

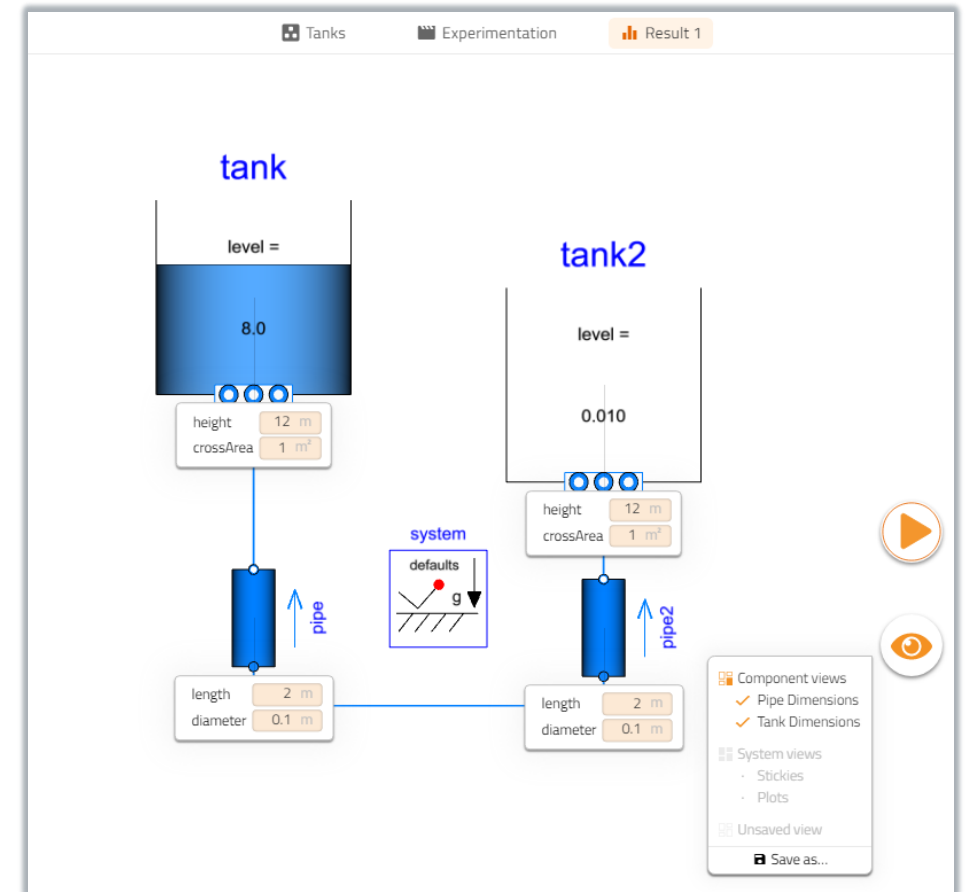
# VIEWS

- Save a view to a model via the -button (press enter)
- At least one sticky or plot needs to be present
- Can combine several saved views



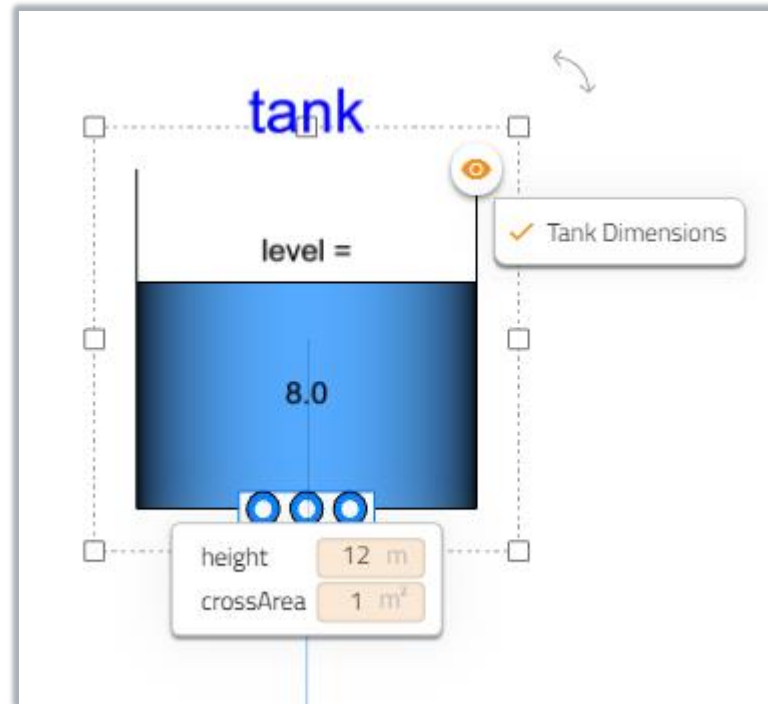
# VIEWS

- Views are saved with the model and can be reused when it is used as a component in a system model
- Referred to as “Component views” in the system model
- Only within workspace
- Works both for global libraries and workspace packages



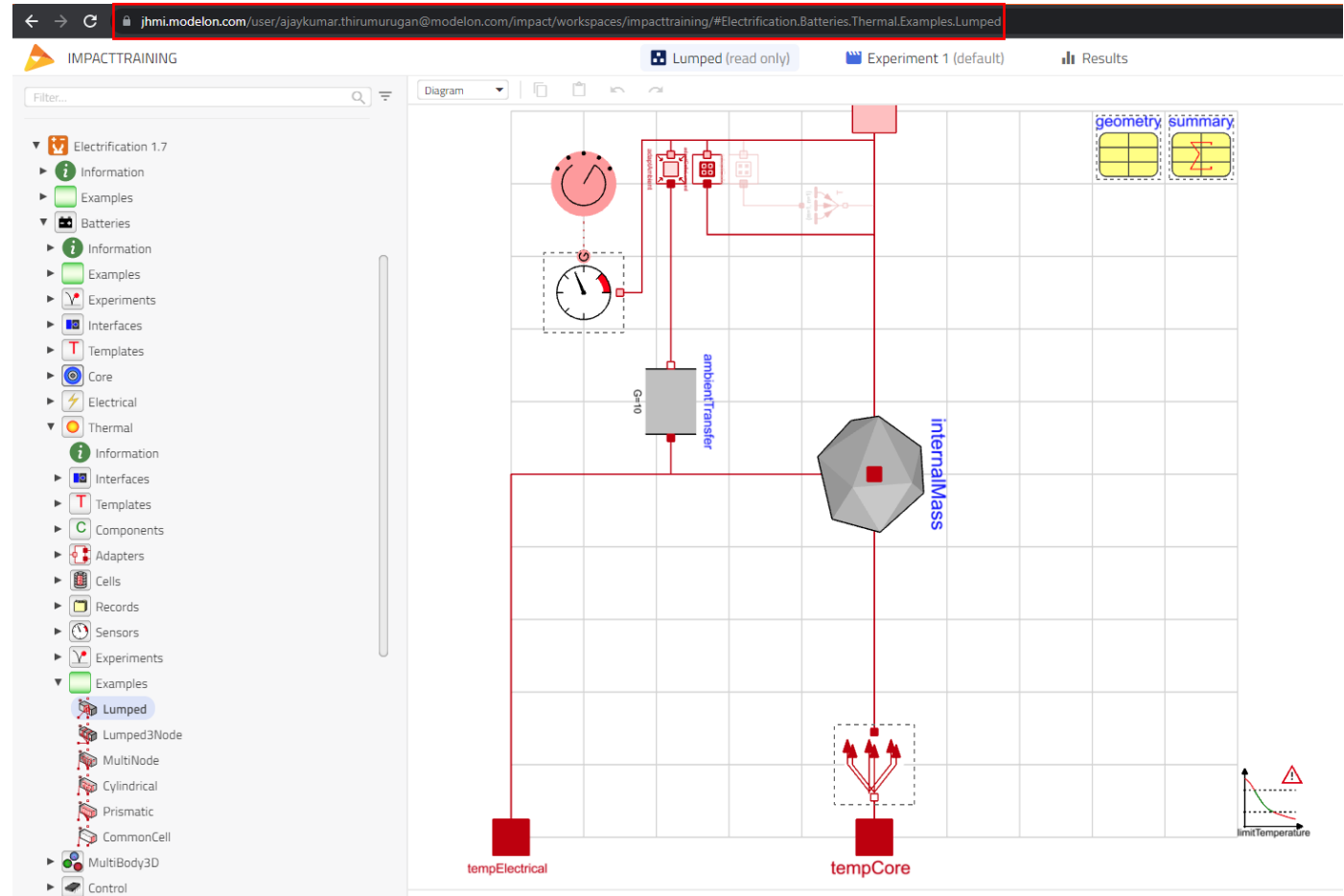
# VIEWS

- Component views can be activated/deactivated directly from the canvas if selected



# VIEWS

- Note that every level in a hierarchical model has its own URL
  - Example shows a link to a sub-component
- Can be shared with colleagues to point them directly towards a specific part of a model
- Combine with views to point users to specific places of interest in a model



# WORKSHOP 1.2

In this workshop you will:

- Build a heating system
- Set boundary conditions
- Simulate
- Create stickies and views