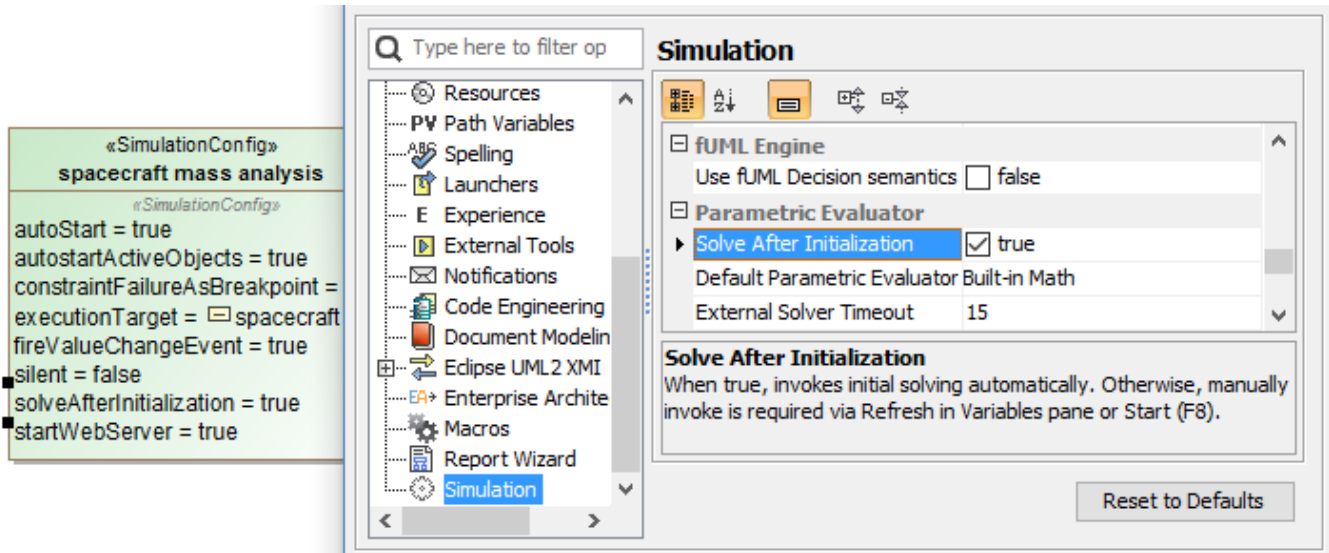


Automatic and manual initialization of objects/values

The **Solve After Initialization** option allows modelers to control initialization of objects and values automatically or manually from the initial parametric constraints solving. This flexibility improves the parametric debugging experience. You can select the option through the Specification window of the Simulation Config, or by clicking **Options > Project > Simulation > Solve After Initialization** as shown in the following figure.

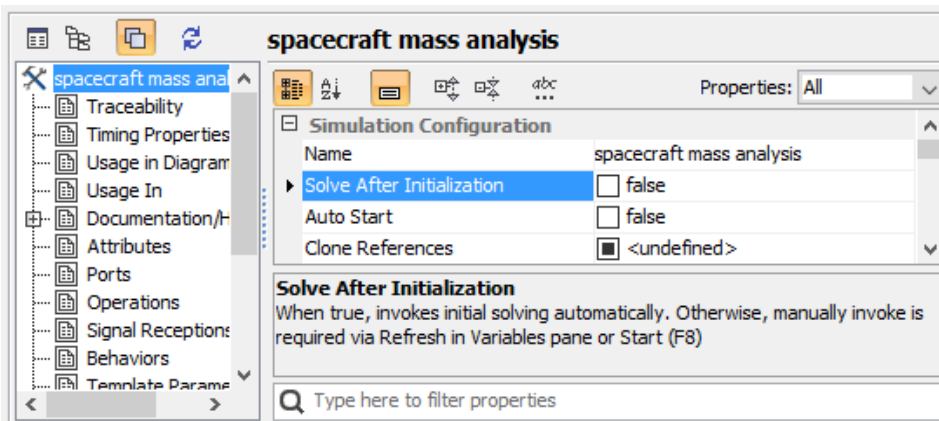


The Solve After Initialization option can initialize objects and values automatically.

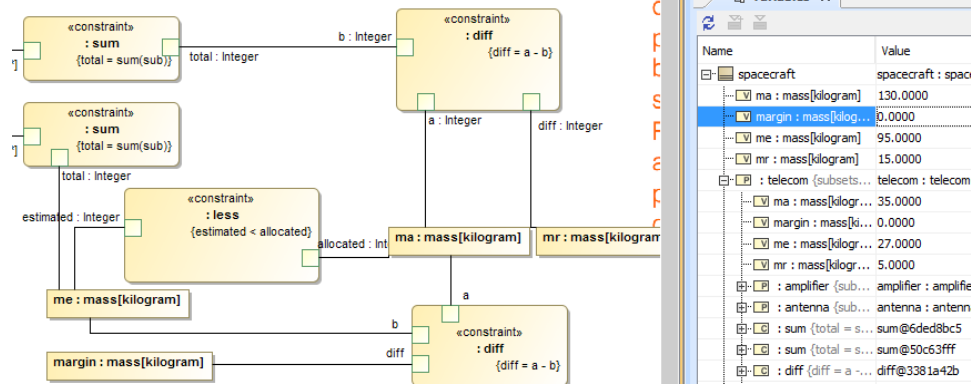
The value of the **Solve After Initialization** option is **true** by default to enable automatic initial solving. If you wish to manually invoke initial solving, you can do so by clearing the check box (the value will change to **false**), clicking **Refresh** in the **Variables** pane, or pressing **Start (F8)**.

To invoke initial parametric solving manually:

1. Open/create a project with a parametric diagram.
2. In the Simulation Config, set the "Solve After Initialization" and "Auto Start" option to **false** (see the following figure).



3. Run simulation via the Simulation Config. All related results will not be calculated automatically, so you need to manually invoke it via the Refresh in Variables pane or Start (F8) (see the following figure).



4. Select **true** for the **Solve After Initialization** option, and run the Simulation Config. All related results will be calculated automatically (see the following figure)

