This course has already ended.

« 1.2 MATLAB and Simulink tutorial | Course materials | 1.4 Feedback »

Running Simulink model with MATLAB script

Main question: How to use MATLAB and Simulink together?
What you are expected to do? Create a simple equation-based Simulink model and run the model with MATLAB script.
Difficulty: Medium.
Approximate work load: 2-3 hours

Start by downloading the following template file. The template zip-file contains necessary files for the exercise of this round.

Download here.

Exercise 2: Vehicle powertrain model

The longitudinal dynamics of a vehicle can be modelled with the following equations:

$$m\frac{dv}{dt} = F_w - F_D - F_{rr}$$

$$F_w = \frac{T_M}{r_d} i$$

$$F_D = \frac{1}{2}\rho c_D A v^2$$

$$F_{rr} = \begin{cases} f_{rr} m g, & \text{if } v > 0 \\ 0, & \text{if } v = 0 \end{cases}$$

Where v is the speed, F_w is the longitudinal force produced by the driven wheels, F_D is the aerodynamic drag force, F_{rr} is the rolling resistance force, T_M is the torque produced by the driven motor. The vehicle features a fixed gear ratio. We only consider velocities $v \geq 0$. Values for the model parameters are given in table 1.

Table 1 Vehicle parameters

Name	Value	Description
m	1150 kg	Mass of the vehicle
r_d	0.28 m	Dynamic radius of the wheels
i	11	Gear ratio
ρ	1.15 kg/m³	Density of air
c_D	0.33	Drag coefficient
A	2.02 m²	Frontal reference area of vehicle
f_{rr}	0.0082	Rolling resistance coefficient
g	9.81 m/s²	Standard acceleration due to gravity

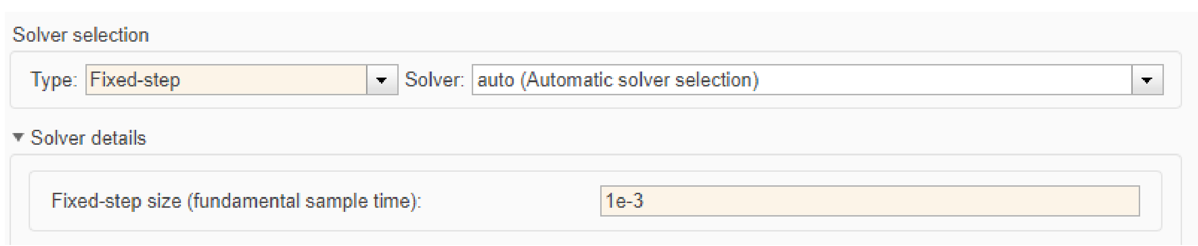
Load data from file

To begin, we will load the velocity profile to the MATLAB workspace from a saved file. The course material contains a file named `velocity.mat`. Open MATLAB and make sure your current directory in MATLAB is in the same directory as the course materials. Double click the file containing the velocity profile. The command window should show a function call to the `load` function (load docs). A new variable should also have appeared in your workspace, named `velocity` and the type should be double timeseries. Plot the velocity profile. Also, save the time at the last timestep of the timeseries in the variable named `t_end`. This will be used as the stoptime in the Simulink model.

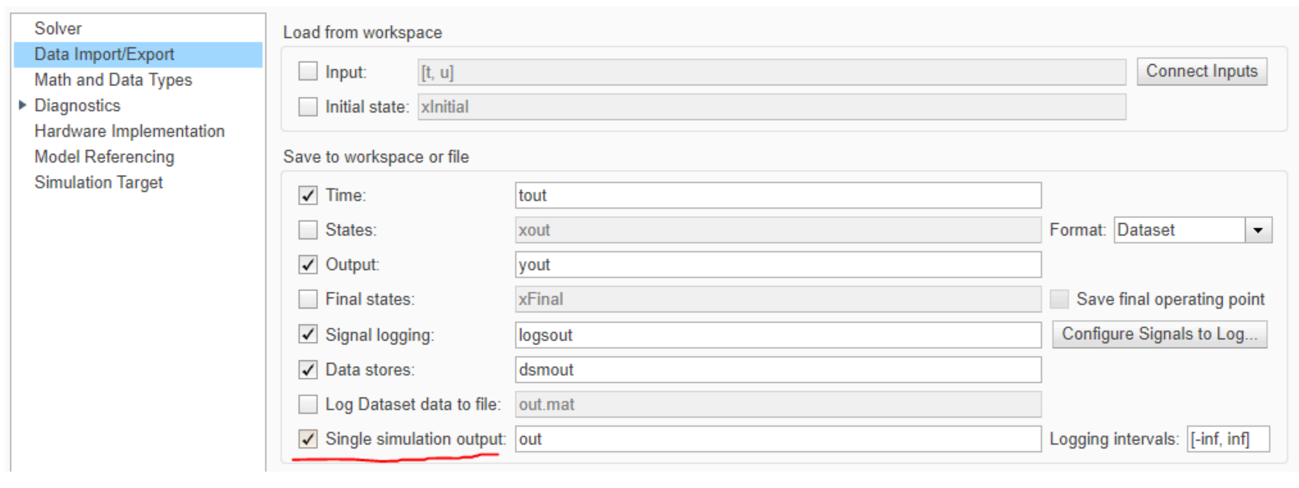
Next, define the given vehicle parameters in your script. Use the names defined in the exercise template. After defining the parameters, you can run the script to initiate the parameters in the MATLAB workspace. This is can be useful for the next step, as Simulink is able to suggest the variables from the workspace when configuring the blocks. By defining the simulation parameters in your MATLAB script, it is easier to change them and for example run the Simulink model repeatedly with different parameters. Save the script, and open a new blank Simulink model.

Create a Simulink model

Open Simulink, select *Blank Model*. Set simulation *Stop Time* to `t_end` from toolbar or in Simulink Model Settings. In Model Settings (`Ctrl` + `E`) under the *Solver tab*, set Solver to *Fixed-step*, and *Fixed-step size* to `1e-3` (=0.001) under solver details shown below.



Under *Data Import/Export* tab, check that the *Single simulation output* output is checked.



Click *Apply* and *OK*. Save the blank model.

These settings Model settings will be used for all future Simulink simulations, to ensure that the A+ autograders can work correctly.

Warning
This basic configuration should be used for all Simulink models submitted in this course. This way we ensure that the automatic grading can work properly.

Hint
If you want, you can save this blank model with the correct settings as a Simulink template. This way you don't have to configure these settings every time you open a new blank model.

1. In the toolbar's **SIMULATION** tab, click the down arrow next to *Save >> Template...*
2. Give your template a Title (for example *MMD_template*) and click *Export*.
3. Now the blank model with the correct settings should be saved under *My Templates*, accessible when you open a new instance of Simulink.

Next start adding blocks to the Simulink model:

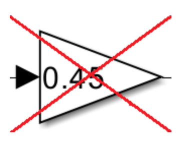
INPUT (Using a From Workspace block)
`v` - Velocity profile timeseries (m/s)

OUTPUT (Using a To Workspace block)
`T_M` - Motor torque (Nm)
`P_M` - Motor power (W)

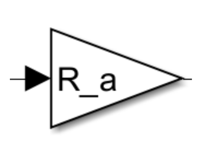
Be sure to use the variable names specified here in the From and To Workspace. Other blocks needed are:

- Constant
- Gain
- Add
- Product
- Square
- Derivative
- Switch (for the rolling resistance)

When configuring the values in the Simulink blocks, use the parameters defined in MATLAB. So, don't use hard-coded values!



But use variables instead:



Simulink sees the variables in the MATLAB workspace, and should suggest them to you in a dropdown. When the model of the system dynamics is complete save it (in the same folder as your MATLAB script).

Running Simulink model from MATLAB

Back in your MATLAB script, run the simulink model with the `sim` function (sim docs). After running the script, and the model should output two new timeseries to the worksapce, `P_M` and `T_M`. Plot the power (in kW. Note: your model output should be in Watts) and torque as subplots, give them titles and label the axes correctly.

Points **0 / 400** My submissions **0 / 10** ▾ Deadline Tuesday, 16 January 2024, 09:00
To be submitted alone

⚠ The deadline for the assignment has passed (Tuesday, 23 January 2024, 09:00).

Using the provided template MATLAB script, load the data into your MATLAB workspace and plot the velocity profile.
Create a Simulink model of the system dynamics.
Plot the results.

Submit the MATLAB script and the Simulink model.

powertrain.m
Choose File No file chosen

powertrain_model.slx
Choose File No file chosen

Submit

« 1.2 MATLAB and Simulink tutorial | Course materials | 1.4 Feedback »