Simulink® Tutorial 2

Simulation of a Resistance Transducer

Aims

- To simulate a resistance type of transducer
- Use a simulation program as a diagnostic and analysis tool to learn transducers

Learning Objectives

Upon completion of this tutorial you will be able to

- To do computation with Simulink
- Use various blocks of Simulink Library Browser
- To edit a Simulink model

Background

The following figure shows a resistance type of transducer that has a linear potentiometer of which total resistance R_T is 10 kΩ and maximum possible displacement X_T is 4 cm. If the maximum power dissipation P is not to exceed 40 mW determine the maximum excitation voltage. Express the output voltage through V, the maximum excitation voltage, and x_i when there is no loading effect.

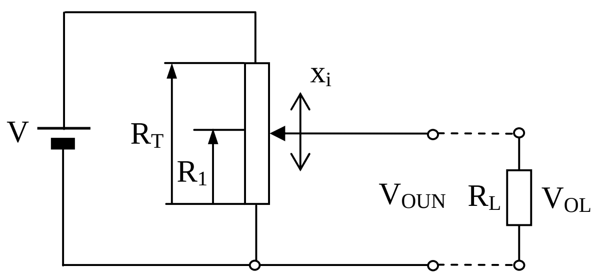

Figure 1 Resistance type of transducer

A meter with a load resistance R_L of 20 kΩ is used to measure the output voltage. Consider the loading effect and calculate the error (%). Make a Simulink model that performs the following tasks:

1. Calculate R_1
2. Calculate unloaded output voltage
3. Calculate loaded output voltage
4. Calculate percentage error

Solution

Mathematical Foundation

The maximum excitation voltage is:

$$V_{max} = \sqrt{PR_T}$$

Thus, $V_{max} = \sqrt{0.04W \times 10000\Omega} = 20V$

This value will be used in the simulation program.

Resistance proportional to x_i (R_1): $R_1 = \frac{x_i}{x_T} R_T$

Then output voltage in unloaded case:

$$v_{OUN} = V \times \frac{R_1}{R_T} = V \frac{x_i}{x_T} \frac{R_T}{R_T} = V \frac{x_i}{x_T}$$

The output voltage in loaded case:

$$v_{OL} = V \left[\frac{x_T}{x_i} + \frac{R_T}{R_L} \left(1 - \frac{x_i}{x_T} \right) \right]^{-1}$$

Error (%):

$$e = \frac{V_{OL} - V_{OUN}}{V_{OUN}} \times 100\%$$

Block diagram algorithm

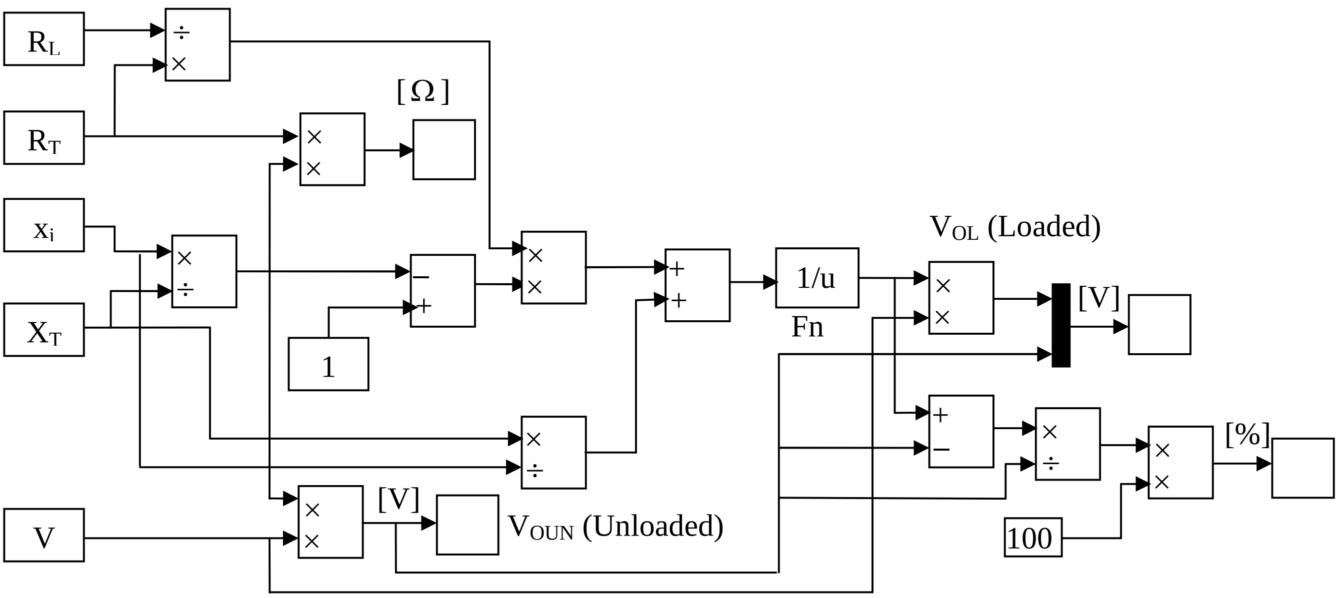

Figure 2 Block diagram algorithm for the simulation program

Hands-on Exercise

Programming

- Open a new Simulink model and save as "ResistanceTransducerSim_Tute_02.mdl"
- Select blocks as in **Fig. 3** from the Simulink Library Browser.

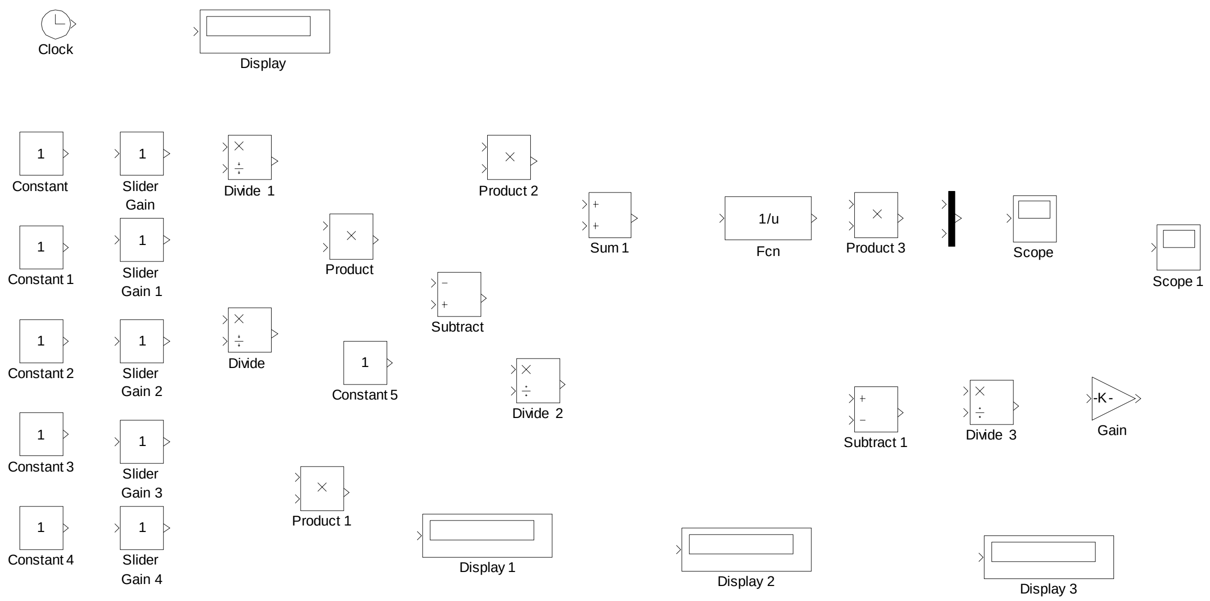

Figure 3 Blocks needed for the simulation program

- Wire blocks properly based on the above block diagram algorithm (see **Fig. 4**)

Edit Simulink Model and Set System Parameters (demo step by step)

- Edit the Simulink model and set system parameters as in **Fig. 4**.

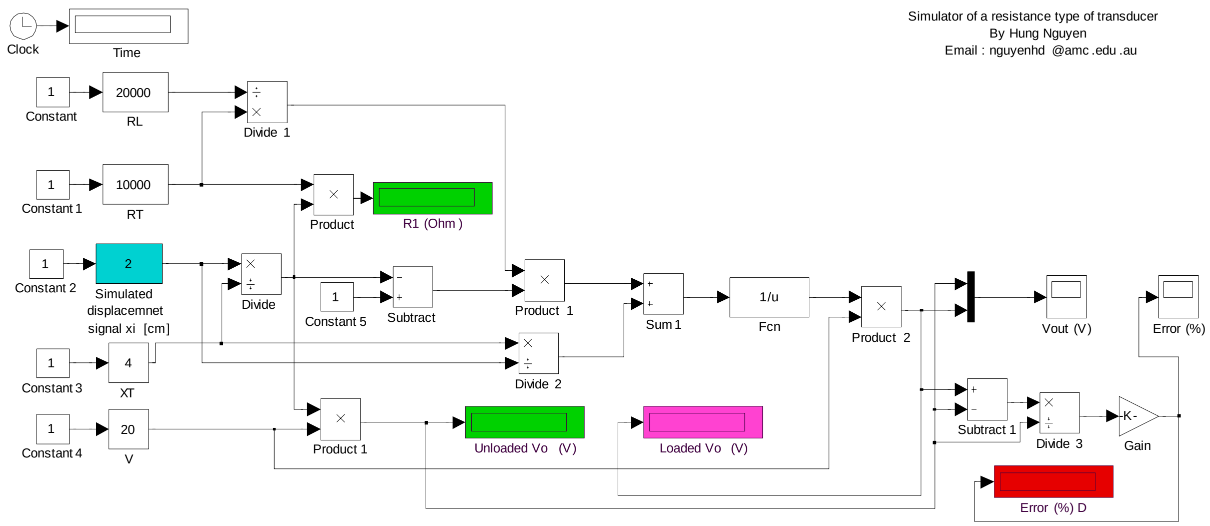

Figure 4 Simulink model after editing and setting system parameters

Configure Simulation Parameters and Test Functionality

- Configure simulation parameters of the Simulink model as in **Fig. 5** below.

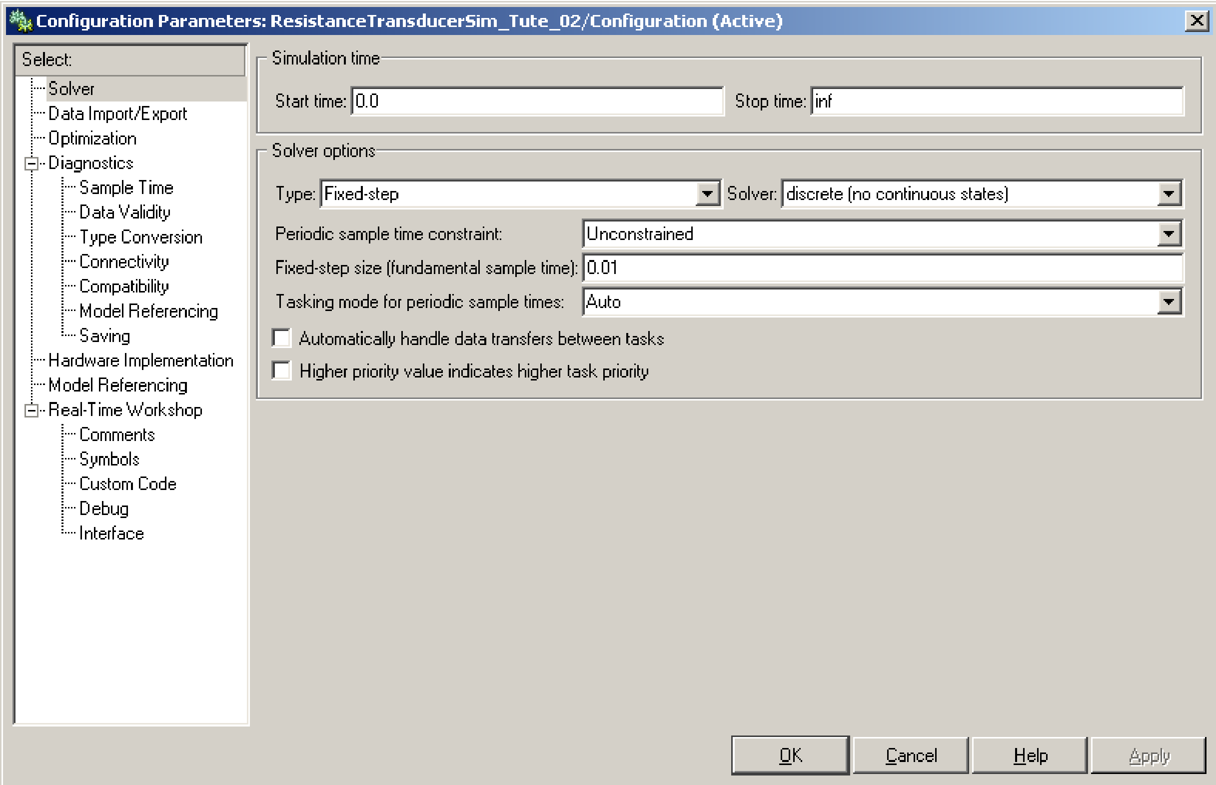

Figure 5 Configuration of simulation parameters

- Save the program.
- Run the program and test its functionality by changing value of x_i.

Use of Simulation Program as a Diagnostic and Analysis Tool

- Run the simulation program
- Investigate how the error changes when changing value of x_i
- Investigate how the error changes when changing value of R_L

Well-done!

Conclusions

At this point the following learning objectives are met:

- To do computation with Simulink
- Use various blocks of Simulink Library Browser
- To edit elements in a Simulink model

Follow-up Exercises

1. Using the above Simulink program (modify the program) confirm the following characteristic of the loaded potentiometer.

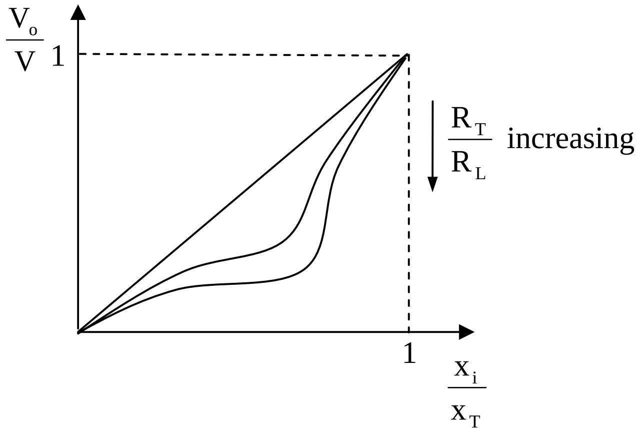

Figure 6 Characteristic of a loaded potentiometer

Hints:

- Use a ramp function to generate x_i (a gradually increasing signal: slope 0.5, 0.0001 to 4)
- Compute V_o/V (V_{OUN}/V, V_{OL}/V) , x_i/x_T and store data to Workspace (you can change value of R_L and the variable name in Workspace block, run the program for couple of times to have different values of R_T/R_L.)
- Plot V_{OUN}/V and V_{OL}/V vs x_i/x_T. The following is an example for 3 different values of R_L:

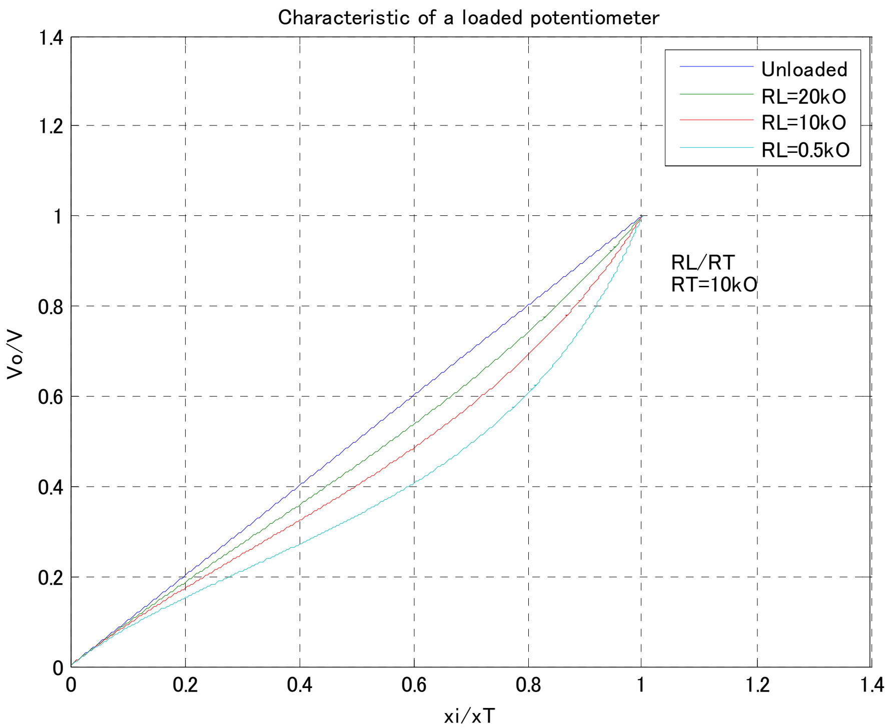

Figure 7 Characteristic of a loaded potentiometer

2. Make a simulation program to simulate an Op Amp as in a non-inverting circuit. Use the simulation program to learn Op Amp. Use these numerical values: V_s = 10 V, R_1 = 1 kΩ and R_F = 5 kΩ (you can change these values by using Slide Gain block to learn the non-inverting Op Amp).

3. Make a simulation program to simulate a Wheatstone bridge circuit. Use these values: R_1 = R_3 = R_4 = 500Ω, R_2 = R_x, V_s = 10 V.