ECE 5671/6671 – Lab 3

Impedance Measurement and Parameter Estimation of a DC Motor

1. Introduction

The objective of this lab is to become more familiar with the hardware and software used in the Electric Generator labs by running a set of experiments using Simulink and dSPACE ControlDesk. The first experiment is to measure the real and reactive powers of a single phase RL load and to calculate its resistance and reactance. The second experiment is to design an open-loop voltage controller to control the speed of a DC generator, operated as a motor. The captured data will then be used to estimate the electric parameters of the DC generator. Note that the two machines (*i.e.,* the squirrel-cage induction generator and the DC generator) **are not** coupled at any point in this experiment.

2. Experiment

Be sure to read the dSPACE DS1104 Control Work Station and Simulink Tutorial (Lab 1) for basic hardware and software setup used in this lab.

Equipment needed:

- dSPACE I/O box
- PEDB with ribbon cable and +12V supply
- Squirrel-cage Induction Generator (SCIG)
- DC Generator (DCG), frame mounted
- Rack of Cables

2.1 Power and Impedance Measurement of an SCIG

For the first part of the lab, you will design a Simulink model to measure the apparent (S), real (P), and reactive (Q) powers consumed by the induction generator at standstill. Consider the following figure for the induction generator winding connection:

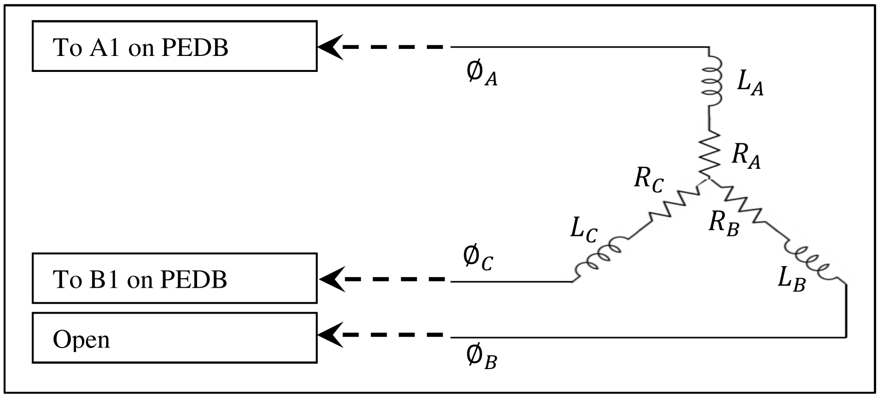

Figure 1: SCIG windings connection

The following variables will be calculated as in Homework 1:

$$V_{RMS} = \sqrt{AVG(v\,(t)^2)} \tag{1}$$

$$I_{RMS} = \sqrt{AVG(i\,(t)^2)} \tag{2}$$

$$S = V_{RMS} * I_{RMS} \tag{3}$$

$$P(t) = v(t) * i(t) \tag{4}$$

$$P = AVG\,(P(t)) \tag{5}$$

$$Q = \sqrt{S^2 - P^2} \tag{6}$$

2.1.1 Simulink model

Start by downloading the **lab_3.mdl** file and **lab_3.lax** layout file provided on the lab website. The Simulink diagram should look like figure 2. Be sure to modify the saturation block and limit the output voltage to 5V in order to prevent possible damage to the SCIG windings. Change the Vd in the gain block named *Ratio* to 42, referring to the 42V DC supply.

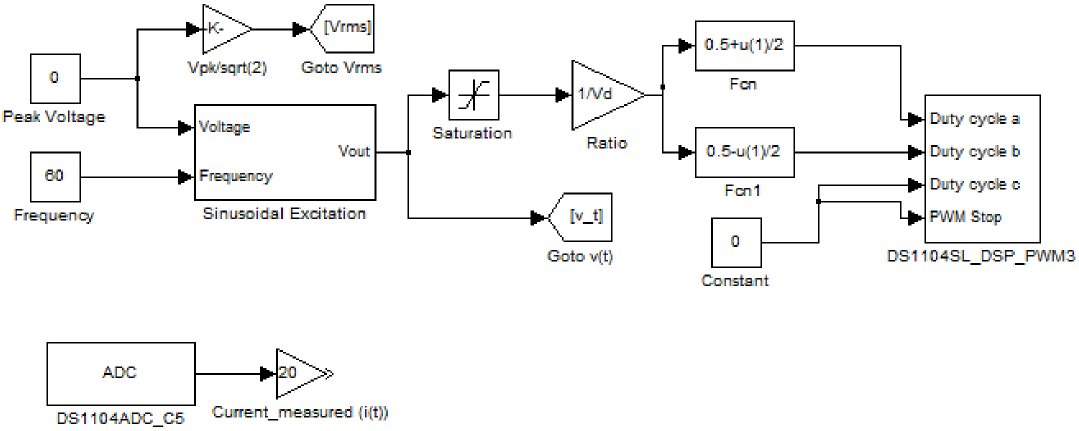

Figure 2: Simulink model to generate AC voltage

Use the analog-to-digital channel to measure the current and place an averaging block with 167 samples (n=167) to find the average current. Do not forget to also change the gain in the averaging block to 1/167. This is similar to the averaging blocks created for current and velocity in lab 1. The averaging length corresponds to one period at 60Hz or two periods at 120Hz.

Follow equations 1-6 to create a Simulink model that calculates the RMS current and apparent, real, and reactive powers. Also, add *Inverter Board Control Function* blocks from the tutorial to the Simulink model to *Stop/Start* and *Reset* the board. Set the sampling time to 100µs by inputting Ts = 1e-4 in the MATLAB *command window*, or entering it in in the initialization function by typing Ts = 1e-4 in **File > Model Properties>>InitFcn**. Check for the correct model configuration parameters. The following should be the standard settings for every Simulink model created henceforth (these can also be found on the checklist file on the lab website, but are provided here for convenience):

> Simulation > Model Configuration Parameters

>> Solver > Start time= 0.0, Stop time= inf, Type = Fixed step, Solver= Ode1 (Euler).

>> Data Import/ Export > Additional parameters > Uncheck 'Limit data points to last: (integer)'.

>> All Parameters tab search for **'Block reduction'. Uncheck** that box. Follow the same procedure to uncheck **'Signal storage reuse'** box**.**

>> Code generation > set 'System target file' to rti1104.tlc by choosing it in the browse options.

It was stated in the lab1 tutorial that the program starts running immediately after being built in the Simulink window. To give the user the capability of starting the program when desired, or ready to do so, the following setting can be modified in the Simulink window

>> Expand Code generation by clicking on the dropdown list > RTI simulation options > ensure that the 'Initial simulation state' is set to 'STOP'.

Review the rest of the settings that are prone to give you build errors, as given in the tutorial handout. Then, build the Simulink model by typing **Ctrl+B**.

Hint: *From/Goto* blocks can be used in Simulink to make signal routing cleaner and easier to read. These are found in the *Library Browser* under *Signal Routing*.

2.1.2 Electric parameter measurement

Before connecting the SCIG to the PEDB, use a multimeter to measure the resistance between two phases. The resistance value will also be measured using an AC current. When the step described below will be completed, compare the two values.

Refer to the tutorial for the hardware setup and connections. Connect any two phases of the SCIG to phases A1 and B1 on the Inverter Board, as shown in figure 1. Also, connect the ADCH 5 on the dSPACE I/O box to the CURR. A1 on the PEDB with a BNC cable. As described in the dSPACE DS1104 Control Work Station and Simulink tutorial, create a new project and experiment. Navigate to the 'Layouting' tab on the top and import the layout previously downloaded. This layout will have radio buttons (with the appropriate values) set up to RUN and STOP through the *simState* (simulation state) variable. The *simState* variable (which can be found amongst the list of variables when the .sdf file is selected in the 'Variables' window') has already been mapped to the radio button block. The RUN button should be engaged after going online, to activate the experiment program. And then, the Start_stop and Reset check boxes will be used to control the relaying of commands to the motor through the PEDB (this is of course when your layout is ready to run the experiment). Modify the layout by adding appropriate instruments to apply a specified peak voltage and display the RMS current and voltage, apparent, real and reactive powers.

Make sure your PEDB is powered as described in the tutorial. Using dSPACE, apply $1V_{PEAK}$ to the windings and measure the line-to-line current and powers at 60Hz frequency. Note that the generator will not be rotating; you are merely measuring the power dissipated within the electrical components of the generator.

Make necessary modifications to the Simulink model to calculate the line-to-line resistance, reactance, and inductance of the SCIG from measured powers. Keep the experiment disengaged and the board unpowered while you make these modifications as this could take a while. If you would like to check your answers in real time, you can add display boxes in dSPACE that will display the values of these or other parameters you would like to view. Record these values in a table.

2.2 Parameter Estimation of DC Motor

In this part of the lab, you will measure the electrical parameters of a DC generator using dSPACE. For this purpose, first design an open-loop controller in Simulink to control the speed of the DC generator. Refer to section **V** of the dSPACE DS1104 Control Work Station and Simulink Tutorial for details about the Simulink model of an open loop voltage controller for a DC machine and dSPACE ControlDesk.

Note that this model will be used in future labs to control the speed of a prime mover to rotate other generators. Understanding this model and how the system works will make the future labs easier. Also, note that throughout this lab the DC generator will be operated as and referred to as a DC motor.

Consider the following model for a DC motor:

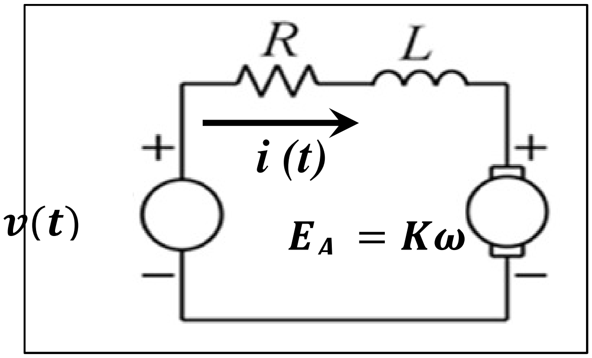

Figure 3: Model of a DC Motor

The mathematical model of DC motor can be described as follows:

$$v = L\frac{di}{dt} + Ri + K\omega \tag{7}$$

where the parameters and their units are as following:

V : $Votage\ applied\ to\ the\ motor\ [V]$

i : $Current\ [A]$

R : $Armature\ resistance\ [\Omega]$

L : $Armature\ inductance\ [H]$

K : $Back - EMF\ constant\ \left[\frac{N.m}{A}\ or\ V.s\right]$

ω : $Angular\ velocity\ of\ the\ motor\ [\frac{rad}{s}]$

In steady-state, equation 7 becomes:

$$v = \ Ri + K\omega \tag{8}$$

This equation will be used to calculate the armature resistance and the back-EMF constant of the DC motor for slowly varying voltages.

2.2.1 Simulink Model

Make necessary adjustment to the Simulink model of the open-loop voltage controller presented in section-V of the dSPACE DS1104 Control Work Station and Simulink Tutorial. Keep in mind to change the delay length to 167 in the averaging block pertaining to the motor current. Make sure to include Stop/Start, Reset and Zero Encoder commands as well as the ability to read the data from the encoder and the current of the motor. Do not forget to set the model configuration parameters as described in section **2.1.1**.

Next, create a new experiment. This time, you will be taught to create the *simState* radio button block. In the layout,

>>Instrument Selector > Standard Instruments > select **Multistate Display** and draw it on the layout. Also, select **On/Off Button** from the same menu.

You will see that a block with two radio buttons will be created. This number can be changed by accessing the instrument properties (this, however, isn't necessary, since you need two radio buttons as you saw in the previous part of this lab).

>> Right click on the **On/Off Button**. Go to **Instrument Properties** and the browse the **Buttons** option just under the **Push Button**.

In the resulting **Buttons** window, you can change the names and values of the buttons.

>> Select **Button 1** > under the properties, enter **‘STOP’** for the *Text* and change the *Value* to **0**.
>> Select **Button 2** > under the properties, enter **‘RUN’** for the *Text* and change the *Value* to **2**.

Now, drag and drop the *simState* variable in the radio button block in the layout, as done before. Design the rest of the ControlDesk layout in dSPACE to take the measurements.

2.2.2 Connection

For the hardware setup and connections, also refer to the dSPACE DS1104 Control Work Station and Simulink Tutorial. Before running the measurement, make sure that the encoder is measuring the position and velocity of the DC motor properly by rotating the DC motor shaft one revolution, 360 degree, and confirming the motor shaft position on the dSPACE ControlDesk. Because the encoder reads a negative value for a positive voltage applied, make sure that the Simulink model inverts the encoder data as done in the lab tutorial. Also, check the analog-to-digital channel, and make sure that it is measuring the correct values. For this purpose, you can follow the analog-to-digital converter and digital-to-analog converter sections in the tutorial.

2.2.3 Measurement

Once the layout is set up, arrange for automatic data export in .mat format. Apply a trigger rule using the Start_Stop variable for a positive edge of 0.5. There should be no need for a trigger delay. Ensure that your duration trigger is set to a value approximately, if not, greater than the time you suppose you will need to record the following set of data. Put the program on RUN mode. Click on *Start Triggered Recording* button on the recorder. Check the *Start_Stop* check button and apply steps of increasing voltage, for example 5, 10, 15, 20, and 25V in succession at approximately regular intervals. Click on the *Stop Recording* button, reduce the voltage and stop the motor. Measure the current, velocity, and position of the DC motor as they change (engage plotters) and tabulate current and velocity for each increment of voltage. Note that the angular velocity unit in the mathematical DC motor model (Eq.7) is in rad/sec. In MATLAB, filter the noise observed in the data if necessary, using a Butterworth filter as follows:

```
[b,a] =butter(3,0.1);
omega_filtered = filtfilt(b,a,omega)
```

This code creates a third-order filter with a cutoff frequency of 500 Hz (0.1/(2*Ts) where 0.1 is the 2nd argument in the function *butter* and Ts (sampling period) is 0.0001 as set in MATLAB. Compare the filtered data with the original; you should notice the reduction of noise.

With the data obtained above, and the steady-state equation (Eq. 8), it is possible to obtain a plot of the Voltage/Current vs Velocity/Current, which should satisfy the following equation:

$$\frac{v}{i} = R + K\frac{\omega}{i} \tag{9}$$

By fitting a line to the plot, derive the back-EMF constant and the armature resistance of the DC motor. Hint: you may use Matlab function POLYFIT to fit the data.

3. Report Requirements:

Use the following as a guideline when preparing the lab report:

- Introduction with objectives
- Include the equipment number of all of the major components used
- Summary of procedures and equations used to calculate the electrical parameters in both experiments
- Provide screen shots of the Simulink models
- Calculated RMS voltage and current, line-to-line real, reactive, and apparent power of the AC induction motor
- Calculated resistance, reactance and inductance values for AC induction motor
- Plots showing the measured current, velocity and position of the DC motor for the different values of voltage.
- Table showing the measured current and velocity of the DC motor for the different values of voltage.
- Plot the Voltage/Current vs. Velocity/Current
- Calculated resistance and back EMF constant for the DC generator
- Conclusion (Describe what worked well and did not work well in this lab, and make suggestions for possible improvements.)