PSpice Simulation of Vibrating Sample Magnetometer Circuitry

Ekta Gupta [1]
[1]*M. Tech Student ECE Department, .Rajiv Gandhi Proudyogiki Vishwavidyalaya, Bhopal (M.P.), India*

Mr. RR Yadav [2]
[2]*Scientific Officer-D, Raja Ramanna Centre of Advance Technology, Indore, India*

Abstract

In this paper, we present computer simulation based study of electronics circuit associated with the sample vibrator which is the subsystem of vibrating sample magnetometer (VSM) using Cadence OrCAD Capture tool for PSpice simulation. The PSpice is an analog/digital circuit simulator which calculates voltage and current (frequency) in a circuit under variety of different circumstances. This feature of PSpice is used to simulate a circuit associate with sample vibrator of VSM. Sample vibrator is an important part of the VSM which is build to fulfil the principle requirements that is pure sinusoidal vibrations of constant frequency and stability of vibration amplitude of the signal. The VSM is suitable for the study of magnetic properties of materials in high magnetic fields. To put the simulation study on firm footing an experimental verification is also carried out in the Lab by developing a PC based data acquisition system.

Keywords-VSM, PSpice, Circuit Simulator, OrCAD Capture, Sample Vibrator, Data Acquisition System.

1. Introduction

A VSM is used to measure the magnetic behaviour of magnetic materials. The Vibrating Sample Magnetometer (VSM) is based upon Faraday's law, according to which an electromotive force (emf) is induced in a conductor by a time-varying magnetic flux. In VSM, a sample magnetized by a homogenous magnetic field is vibrated sinusoidally at small fixed amplitude with respect to stationary pick-up coils. The resulting field change inside the pick-up coils (detection coils), induces voltage and from measurement of this voltage the magnetic properties of sample deduced [1]. A second voltage is induced in a similar set of reference coil by a reference sample which may be a small permanent magnet an electromagnet. Since the sample and reference are driven synchronously by a common member, the phase and amplitude of the resulting voltages are directly related. The known portion of the voltage from reference coil, phased to balance the voltage from detection coil, is then proportional to the magnetic moment of the sample. By this procedure the measurements can be made insensitive to changes of vibration amplitude, vibration frequency, small magnetic field instabilities, magnetic field nonuniformity, amplifier gain, or amplifier linearity. So the associated electronic circuits with sample vibrator serve the function of a null detector [2, 9].

Schematic diagram of a vibrating sample magnetometer shows in figure 1 [5]. Where vibration unit or sample vibrator consists reference coil is used to provide vibration at constant frequency to the sample holder. Sample is fixed to the end of the sample holder.

The sample vibrator is main subsystem of VSM which is built to fulfil the following principle requirements: (i) Pure sinusoidal vibrations of constant frequency; (ii) Stability of vibration amplitude under load variation and friction. For controlling the vibrator an electronic circuit has been designed [1].

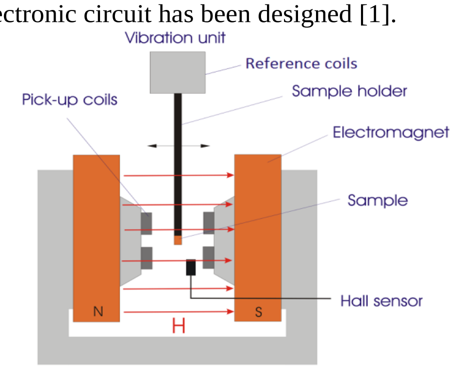


Fig.1 Schematic diagram of a vibrating sample magnetometer

Different types of magnetometers have been developed and are now commercially available .The first VSM was designed and constructed by Professor Simon Forner of the Massachusetts Institute of Technology, USA [2]. Since then, vastly improved versions of VSMs have been developed by various manufactures and are now commercially available. They have been extensively reviewed by Foner [3, 4]. These all types of magnetometer consists electronic system which is manually operated and also required lots of time and cost for fabrication of different type of electronic component of the circuit. Therefore, much attention was paid to make design and simulate the circuitry associated with VSM for controlling the frequency and amplitude of the sample vibrator [1] by using OrCAD capture tool [8].

2. Electronic Circuit Associate with Sample Vibrator

The functions of the associated electronic circuits are: (1) to permit accurate calibration of the signal output obtained from the detection coils, (2) to produce a desire sine wave signal with 20-170 Hz directly related to the input and, (3) to provide sufficient amplification for high sensitivity operation. These functions can be performed by a variety of circuits. The block diagram of the vibrator electronic circuit is shown in figure 2. Description of individual parts and their features are given below:

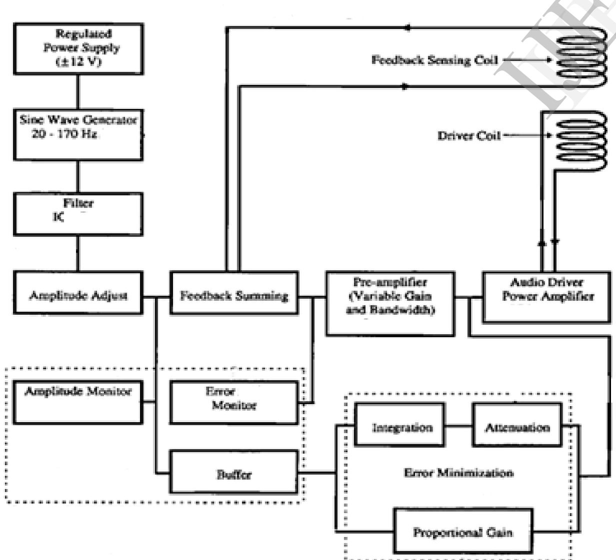


Fig. 2 Circuit Block Diagram

2.1 Sine- wave Generator

. It requires adjusting variable frequency range between 20-170 Hz. The output of sine wave generator is feed into the sine wave filters for phase shifting and to obtain desired shape

2.2 Error Monitor, Amplitude Monitor and Buffer

Here error monitor is used to correct the error by using voltage divider circuit to reduce the voltage obtained from pickup coils. Amplitude monitor act as clipping circuit to adjust the amplitude of the waveform obtained from the pickup coils. Buffer is used to reduce the loading effect between two devices and provide proper shape of sine wave.

2.3 Error Minimization Circuit

The error minimization facility [1] is a variable voltage feed which can be applied directly to the power amplifier. This has an in-phase (proportional) component and also a phase-lagging (integral) component with respect to the reference voltage (chosen velocity). In this way a substantial reduction in the difference (error) between the required velocity obtained through sensing coil(pick- up coil) and that obtained through the reference coil of the driver can be achieved. This feature is essential in order to operate the driver coil very near to the reference velocity irrespective of variations in load and frictional changes in the vibrator and sample support. This allows one to calibrate the magnetometer at a fixed reference velocity using standard samples of known magnetization. The same vibrating conditions can then be used for the magnetization measurements of unknown samples.

2.4 Preamp and Power Amplifier

A preamplifier is used as a high gain amplifier to prepare a small electrical signal for further amplification or processing and reduce the effects of noise and interference. It is used to boost the signal strength to drive the cable to the main instrument without significantly degrading the signal-to-noise ratio (SNR). The second amplifier is typically a power amplifier. The preamplifier provides voltage gain (e.g. from 10 milli volts to 1 volt) but no significant current gain. The power amplifier provides the higher current necessary to drive loudspeakers attach through reference coil or driver coil which is connected to sample vibrator to provide vibration.

3. Simulation Software

SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis and was inspired by the need to accurately model devices used in integrated circuit design. It is a general purpose software that simulates different circuits and can perform various analysis of electrical and electronic circuits including time domain response, small signal frequency response, total power dissipation, determination of nodal voltages and branch current in a circuit, transient analysis, determination of operating point of transistors, determinations of transfer functions etc. This software is designed in such a way so that it can simulate different circuit operations involving transistors, operational amplifiers (op–amp) etc. and contains models for circuit elements (passive as well as active) [8].

It was first developed in the University of California, Berkeley, USA. Subsequently an improved version SPICE 2 was available now progressed to SPICE3 to support computer aided designs. PSpice is the member of SPICE family and it is a commercial software product based on SPICE algorithm. It is useful for simulating all types of circuits in a variety of applications. The increased utilization of PCs has led to the production of PSpice, a widely available PC version distributed by the MicroSim Corporation whilst HSPICE from Meta-Software has been popular for workstations and is now also available for the PC. PSpice has become one of the most popular circuit simulation programs now marketed by OrCAD [7], which can draw the circuit and create a schematic file. The availability and the capability to share its evaluation version freely at no cost is one of the reasons for the popularity of Pspice.

By using PSpice software traditionally, electronic circuit design was verified by building prototypes, subjecting the circuit to various stimuli and then measuring its response using appropriate laboratory equipments. Prototype building is somewhat time consuming, but produces practical experience from which we judge the manufacturability of the design. Computer programs that simulate the performance of an electronic circuit provide a simple cost effective means of confirming the intended operation prior to circuit construction and of verifying new ideas that could lead to improve circuit performance. Such computer program like OrCAD capture combine with schematic design capture technology, with extensive simulation and board layout technology, Cadence [8] help you capture design intent correctly the first time. OrCAD capture CIS [8] integrate the features of a component information system (CIS) with Cadence schematic capture technology. These products are designed to reduce production cost overruns through efficient management of component. The time spent searching existing parts for reuse, manually entering part information content, and maintaining component data is reduced. Users search for parts based on their electrical characteristics and CIS automatically retrieves the associated parts. The easy-to- use technologies allow designer to focus their creativity on design rather than tool operation. In this paper we had performed the OrCAD capture CIS, transient analysis of the elaborated electronic circuitry of VSM.

4. Schematic Design Entry And Simulation Report

The prototype of VSM is an in- house designed and constructed sample vibrator. To control the vibratory frequency and amplitude we have designd an electronic circuit using OrCAD capture tool to generate sine wave of desired frequency and amplitude. PSPICE based simulation and schematic design of individual parts of electronic circuitry for controlling the vibrator is shown below where corresponding input and output voltages are show by different waveforms. The circuit specifications for figure 3 which is designed by combining weignbridge sine wave generator which generates input AC Voltage=20V, frequency=45Hz and load resistor=1Ω and sine wave filter for getting desired shape of sine wave. Error monitor shown in figure 4 which act as a voltage divider circuit, which reduces the voltage from 1 volt to 0.1 from pick up coil for amplitude adjustment of vibration. Then output of error monitor circuit feed to amplitude monitor shown in figure 5 which is used to reduce the loading effect between two devices and provide proper shape of sine wave of same frequency. Pre and power amplifier shown in figure 7 by cascading two opamps to first one is pre amplifier which provide gain from 1volt to 2 volt and second one act as power amplifier which provide higher current necessary to drive loudspeaker of same frequency of 50 Hz.

.

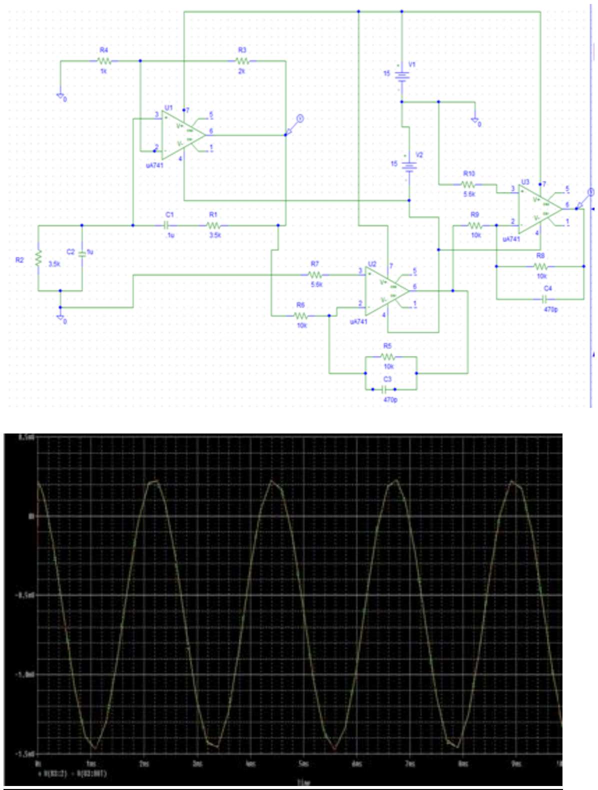


Fig 3 Schematic Design and simulated waveform of sine wave filter

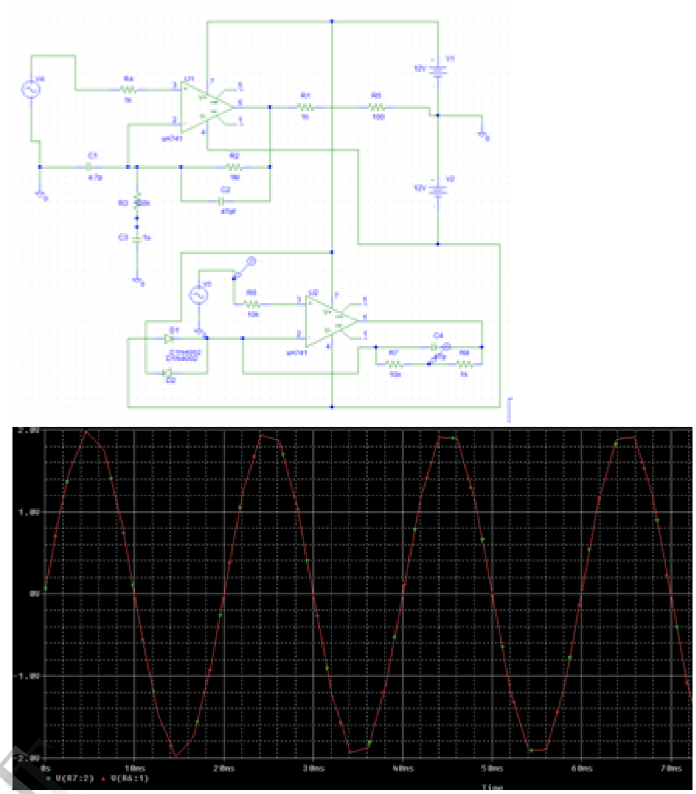


Fig 5 Schematic Design and Simulated waveform of Error monitor and amplitude monitor

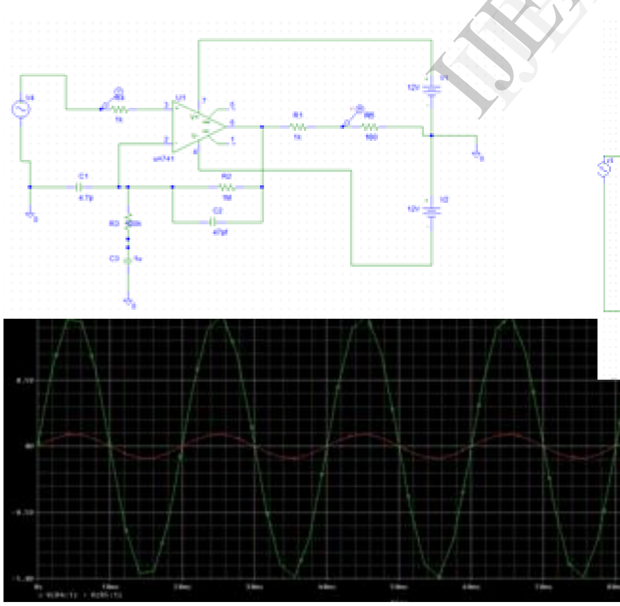
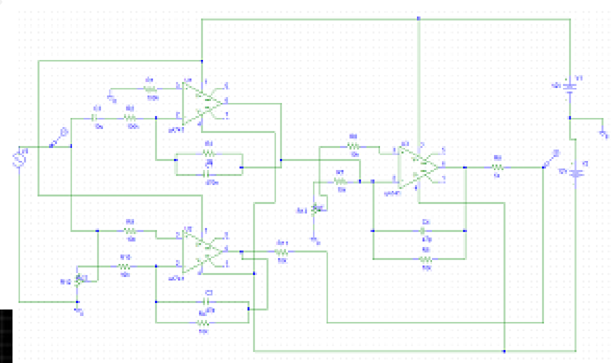


Fig 4 Schematic Design and Simulated waveform of Error monitor



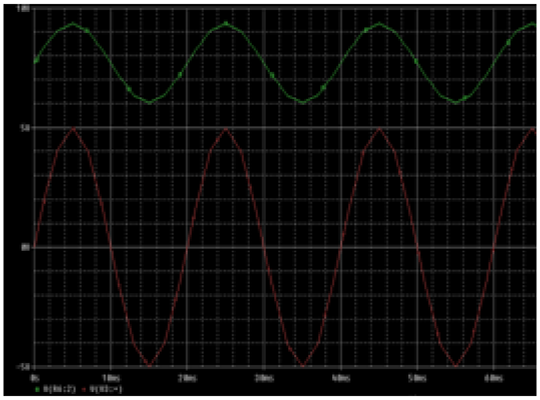


Fig 6 Schematic design and Simulated waveform of Error Minimization Circuit

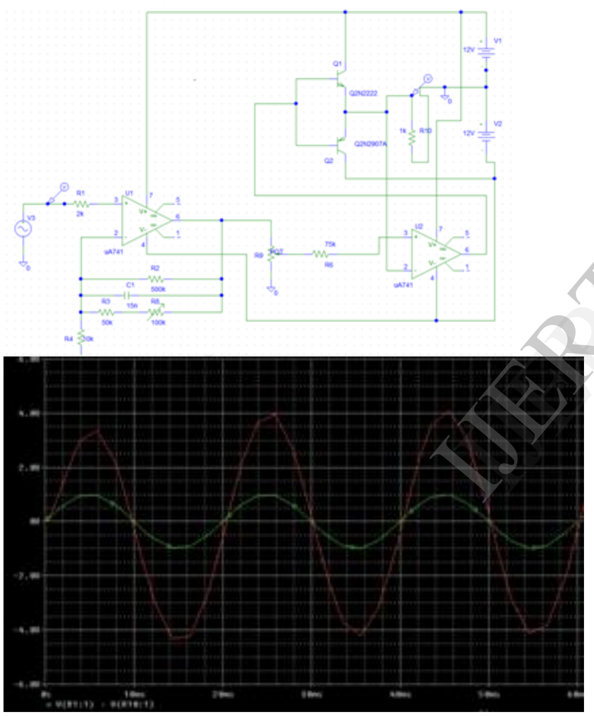


Fig7 Schematic design and Simulated Waveform of pre-power amplifier

5. Conclusion

A complex simulation setup for the design of electronic circuit to control the sample vibrator which is the part of the VSM is proposed. The environment is built around a Vibrating Sample Magnetometer and includes the PSpice simulation for the circuit analysis. The main benefit of the proposed approach is that OrCAD Capture provides the user the possibility to include various design specifications, including fast and intuitive schematic design entry for analog simulation using PSpice and also non electrical ones, such as costs, size etc. A case study to illustrate the efficiency of the setup was presented and the results obtained so far are encouraging, especially taking into account the fact that the execution time proved not to be prohibitive. Experimental observational testing of various modules such as electronic circuitry, induced voltage, actuator frequency and amplitude are found to close agreement with our theoretical value with some percentage variation. The error detection and correction circuit is incorporated to reduce the error due to the reference signal.

6. Acnowledgement

The authors wish to acknowledge Shree R S Sindhe, Scientific Officer-H (Head, AMTD) for providing me an opportunity of working in Accelerator Magnet Technology Division lab of RRCAT and encouraging throughout the whole work. He gave me lot of insight into the subject matter, point out new direction for me and devotes lot of his valuable time for my project work. We thank the Department of Atomic Energy, Raja Ramanna Center for Advances Technology Indore for the financial support.

7. References

[1]. A. Niazi, P. Poddar and A. K. Rastogi, A precision, low-cost vibrating sample magnetometer, Current Science, Vol. 79, No. 1, 10 July 2000.

[2]. S. Forner, Versatile and sensitive vibrating sample magnetometer: Lincoln Laboratory, Massachusetts Institute of Technology, Lexington, MA (1959). .

[3]. Foner, S., *IEEE Trans. Magn.*, 1981, **17**, 3358–3363.

[4]. Foner, S., *J. Appl. Phys.*, 1996, **79**, 4740–4745.

[5]. D. Speliotis. Getting the most from your vibrating sample magnetometer,ADE Technologies Inc.,Newton MA, U.S.A., 43.

[6]. S.Foner, Versatile and sensitive vibrating-sample magnetometer. Rev. Sci.Inst, 30:548, 1995.

[7]. Muhammad H. Rashid, publications, ‘‘Introduction to PSPICE Using ORCAD for circuit and electronics’’.

[8]. http://www.cadence.com/us/pages/default.aspx

[9].S. Foner, Further improvements in vibrating sample magnetometer sensitivity. Review scientific Instrument. 46, 1425-1426(1975)