PSPICE SIMULATIONS FOR PERFORMANCE TESTING OF SIGNAL CONDITIONING CIRCUIT FOR TEMPERATURE SENSOR

A. V. MANCHARKAR

Head, Dept. of physics, New Arts, Commerce and Science College, Parner 414302, INDIA

Email: mancharkar_av@rediffmail.com

Abstract - This paper addresses performance of signal conditioning circuit for temperature sensor viz. J-type thermocouple, as an example. In reality the circuit has to simulate over the specified operation range. The output voltages are up to 3-digits decimal point by experimenting several input voltage values in the test circuit. The various input voltage values are used in performing DC analysis. The low cost, fast simulation techniques for the linearity and an accuracy of the circuit performance and reliability is reported by using the PSpice test tools. The results are explored with browsing output data facility and run probe analysis.

Keywords: **SCC**: Signal conditioning circuit, DC analysis, simulation.

1. Introduction

The evolution of process control has seen the infusion of electronics technology into almost every facet because of low cost, reliability, miniaturization, and ease of interface. Using the signal conditioning circuits the direct transudation of any physical signal is converted into the required output signal. The specific type of signal conditioning depends, of course on the type of sensor employed as well as the nature of the specified output signal characteristic [1]. The electronic industry is getting progressively more and more efficiently at developing new products in wide ranges and variety of sensor applications to customer. We also see more and more products coming into the market with shorter and shorter product lives or some time they may fatal at times [2]. Hence, low cost circuit design, with an accurate, linear and faster testing technique is addressed.

In the industrial processes temperature is a main factor. A variety of temperature sensors are commercially available however signal conditioning plays an important role in design and development of accurate circuit performance and optimum reliability. Here, the product has to be practically designed first and then tested for its intended results, but any degradation in the output signal is found circuit parameters and values have to be replaced and circuit is re-tested for improved performance and low drift in specified parameters over the operating range. To complete one such process involves large amount of time, higher cost and accurate component values. This critical problem of product design and testing is simplified by the use of PSpice simulations. This allows user to design, test, and perform various analysis to optimize the accuracy, circuit performance and its reliability before any product is actually made and transferred to the customer for their use [3]. The signal conditioning circuit for J-type thermocouple temperature sensor is discussed in this paper.

2. Simulation concepts

Electronic Simulation of circuit function is now a common practice in the design both of individual circuits and complete systems. The more of the circuit a designer can simulate, the faster the circuit can get into production and hence to market. PSPICE (PC Version, Simulation Program with Integrated circuit Emphasis) has become a common tool for analogue simulations and widely used, even for some mixed mode circuit designs [4].

Sophisticated instrumentation systems rely heavily on the application of a wide range of electronic circuits. Signal processing circuits constitute the essential link between the sensors and the final output equipment such as readout systems, computers and other devices. The signals obtained from sensors/transducers may be in analog or digital domain, very few instrumentation systems are fully digital by nature. Signal processing and conditioning is carried out in order to bring out the output signal to the desired description and standardized levels. In most cases, it is essential to see that the signal processing circuitry chosen preserves the desired functional relationship between the input and output signals and does not in any way impair the basic accuracy with which the measurement is carried out. The signal conditioner can vary in complexity with or without detectors, and filters. Alternately, they are termed as signal modifiers or signal processors. The output signal may be an analog or digital quantity [1].

PSPICE allow the designer to construct entirely new circuit without fabricating the actual circuit using hard wires, components and PCBs. The circuit performance and its reliability in any circuit to minimize failure can be tested. To meet the required standards of a circuit and hence quality instrument, the circuit analysis is performed. In case of any failure or problems observed, one can easily redesign it by modifying the very same circuit in few minute using highly sophisticated simulation tools. With adequate number of design and redesign iterations on a computer platform where it consumes only a small amount of time and no material. The design can be made robust. The simulated circuit can then be subjected to different analysis *i.e.* actual tests. The performance and reliability of circuit and instrument definitely shows results of up most levels. Thus it is the faster and low cost and less cumbersome process .

Dr. Lawrence Negal at University of California, Berkeley, originally developed PSPICE, It is a general- purpose circuit simulator program that simulates electronic circuits. It performs various types of analysis of electronic circuits. PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used both in industries and universities [5].

To test and analyze the circuit performance first set the circuit for initial conditions such as zero and span adjustments using pots PT2 and PT1 as shown in the circuit diagram shown in Fig.1. Spice simulations and testing process is as follows.

- Drawing the circuit
- Selecting the type of analysis
- Simulation of the circuit
- Displaying the results of the simulation

The Design Center software package has three major interactive programs: *Schematics, Spice,* and *Probe.* Schematics are a powerful program that let's you build circuits by drawing in a window on the screen. Spice analyzes the circuits created by schematics and generates voltage and current solutions. Probe is a graphics post-processor that allows you to display plots of parameters such as voltage, current, impedance, and power [6].

3. Signal Conditioning Circuits

        Measurement systems handle various types of signals produced by the sensors. These signals are required to be conditioned for user friendly access. Following section describes the signal conditioning circuit used for widely used temperature sensor viz. J-type thermocouple.

3.1 Signal conditioning card for Thermocouple

        The J-type thermocouple is used frequently in industry because it is economical and has high output in millivolts over its temperature range. Due to relatively low voltage output associated with most thermocouples, amplification circuit is used to increase sensitivity. The resulting output of this circuit is used to drive or activate a readout device [7]. This card converts the change in thermocouple voltage in mV into the corresponding change in output voltage in mV.

        Figure1: shows a signal-conditioning card for a J type thermocouple. In this card, a +12 V is used to apply excitation voltage voltage. Resistors R4,R5,R6 and capacitors C3,C4 forms a low pass filter. An Op-amp is used in non-inverting mode.
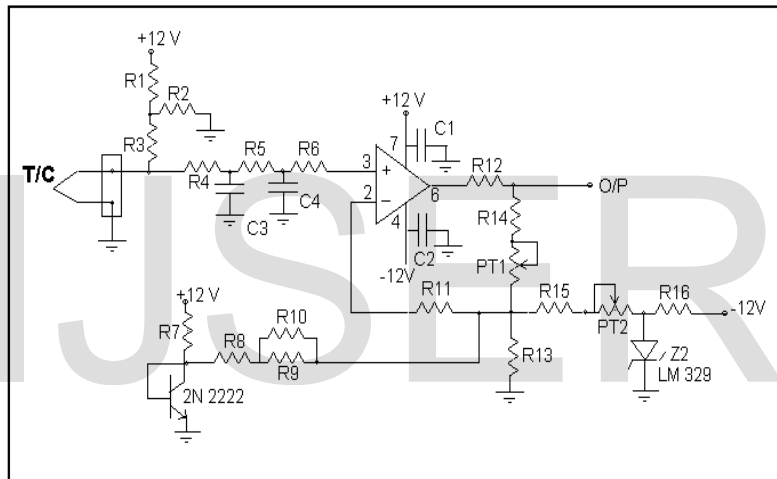

Fig.1 *Signal Conditioning Circuit for Thermocouple*

    For testing the card using PSpice test tools the DC input voltage was swept in the range 0mV to 6mV in step of 0.1mV. While testing the card practically, initially the inputs of card were shorted to the ground and PT2 is adjusted to get a voltage in mV, which corresponds to room temperature. For testing this card, using potential divider an input voltage in mV was applied to the card and output voltage was noted.

4. Results and Discussions

   The PSpice simulation of typical temperature sensor signal conditioning circuit was also carried out successfully. The performance dependence characteristic on input voltage in mVs is shown in Figure2.
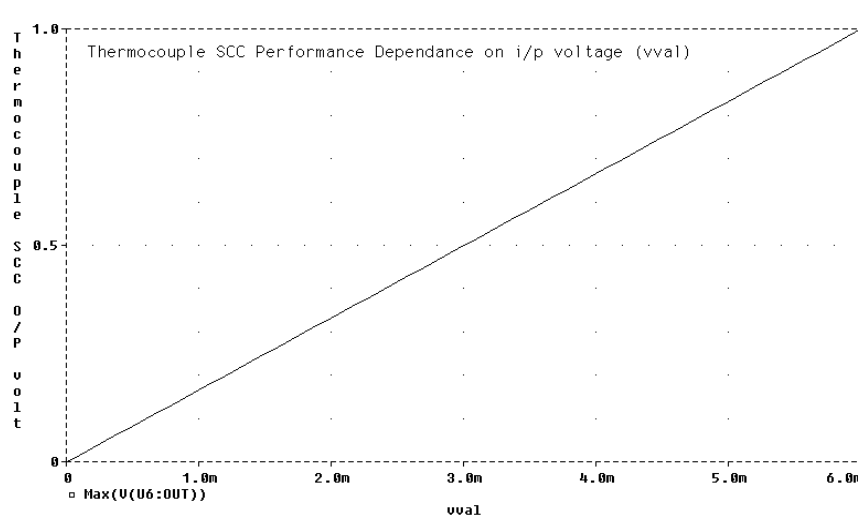
Fig.2

Figure3: shows variation of output voltage with respect to changes in input voltage in mV which is practically applied. It shows that as input voltage increases the output voltage also increases and this change is linear.
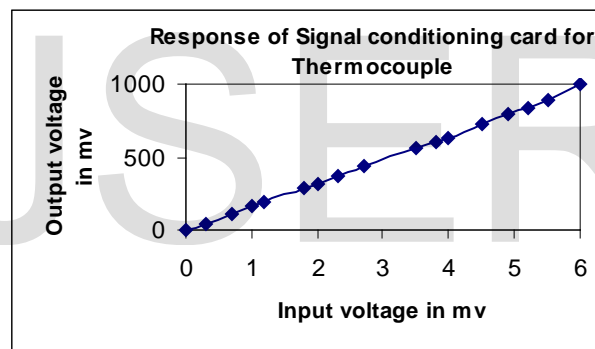

Fig.3

This paper reports that the results obtained after applying inputs to SCC using PSpice test tools and the results obtained after applying inputs practically are same. Since Spice is the industry standard for electronic circuits simulations, it is used extensively for the design of other sensor signal conditioning circuits such as pressure, flow, displacement, force, vibration, etc. in sensor design and applications as the new product. Also these simulation studies are uniformly may be applied for testing the circuit performance and reliability under changing physical conditions under which they are operating before actual circuit or product is made[8]. Hence it avoids the complex, cumbersome, cyclic testing procedures for a newly developed product.

References
[1] Murty D V S, *Transducers And Instrumentation,* Printice Hall of India, New Delhi : 1995.
[2] Ward A E, Angus A S, *Electronic Product Design*, Chapman & Hall Oxford : 1996.

[3] Walunj A K, Mancharkar A V and Shaligram A D, *PSpice Simulations for performance and reliability testing of sensor signal conditioning circuits*, J. Instrument. Soc. India **30** (20) :65-74.

[4] Martin O′ Hara, *Modelling board-level DC-DC converters in SPICE* (Electronic Product Design), July : 1998.

[5] Tunenga Paul W, *SPICE: A guide to circuit simulation and analysis using PSPICE*, Englewood Cliffs N.J. USA,Prientice Hall : 1998.

[6] Rashid M H, *Microelectronic Circuits Analysis And Design,* PWS Publishing Co : 1999.

[7] Rangan C S, Sharma G R, Mani V S V, *Instrumentation Devices and Systems*, Tata-McGraw-Hill Ltd, New Delhi :1997.

[8] Walunj A K, Mancharkar A V and Shaligram A D, *Temperature Sensors SCC PSpice performance and reliability testing due to resistive tolerances and effects,* Proceedings of 7th NSPTS, University of Pune, 14-16Feb, 2000, p50.1-50.6.