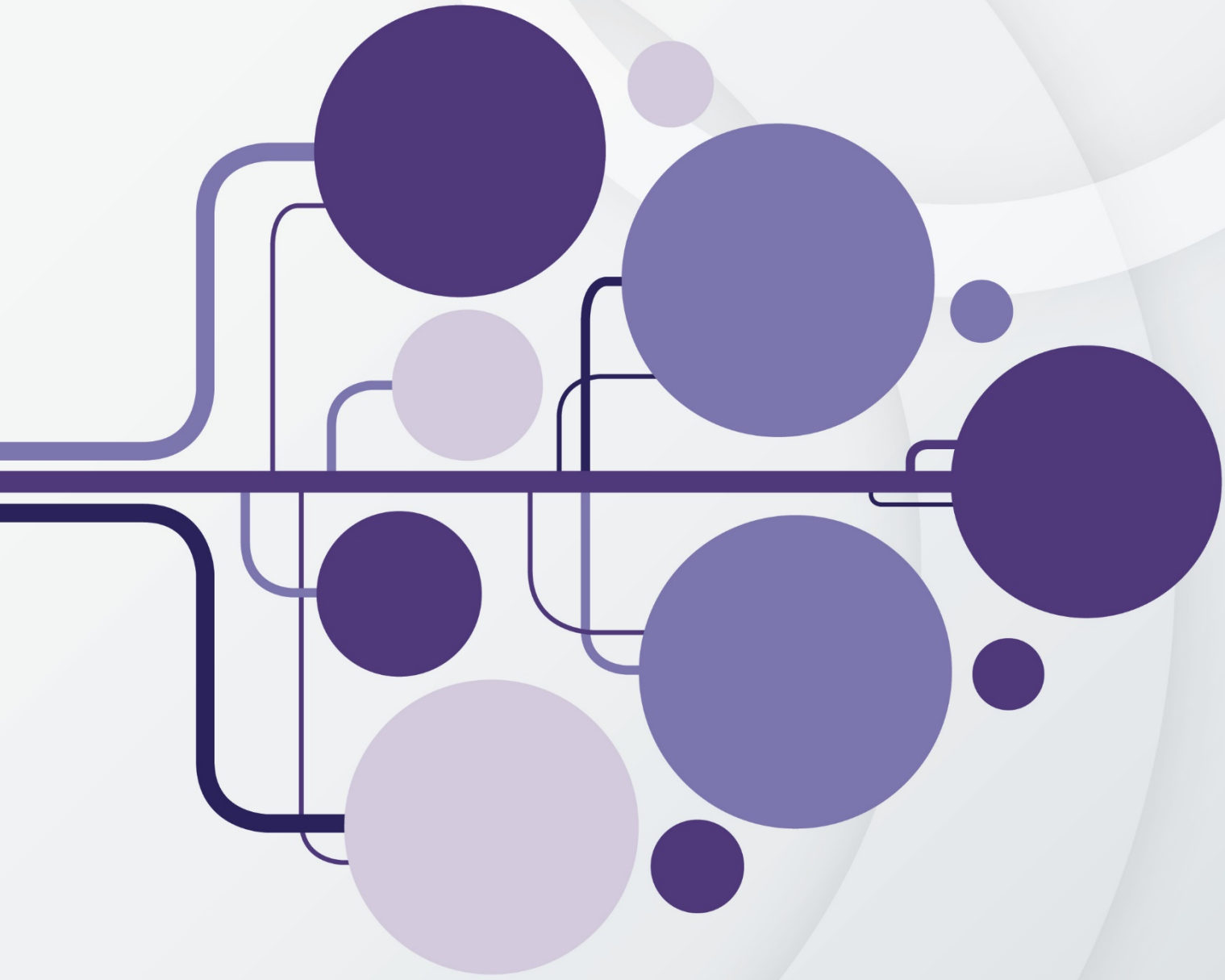
Tekscan

OEM DEVELOPMENT KIT

Version 1.0

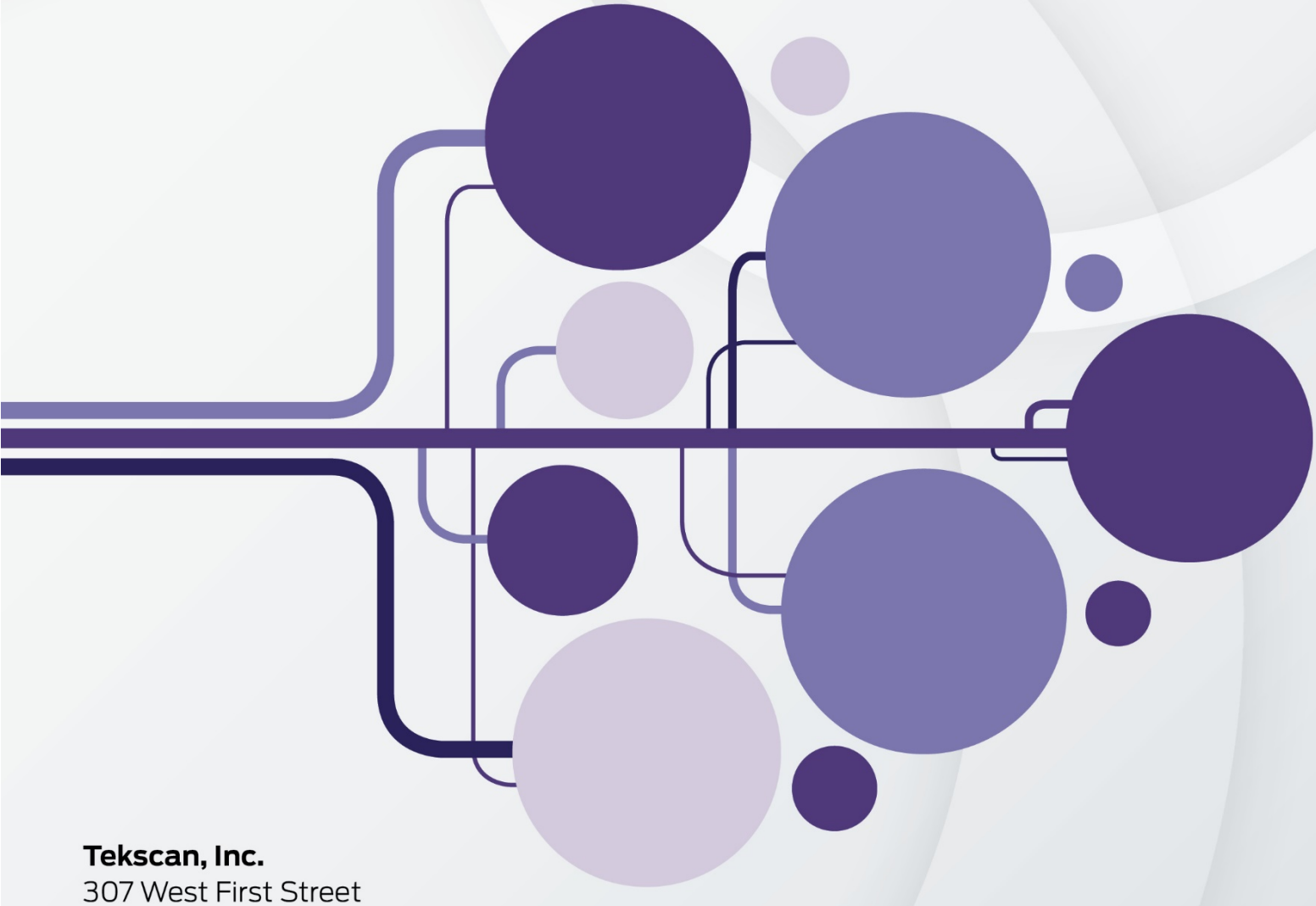


+1.617.464.4283 1.800.248.3669 info@tekscan.com www.tekscan.com/es

Tekscan™

OEM DEVELOPMENT KIT

Version 1.0



Tekscan, Inc.
307 West First Street
South Boston, MA 02127-1309 USA

Support@tekscan.com
Fax: 1.617.464.4266

TABLE OF CONTENTS

Table of Contents 3
Introduction 4
OEM Development Kit Hardware 4
  Components List 4
  The FlexiForce™ OEM Development Board 5
  Specifications 5
  Signal Conditioning Circuit 6
  Sensitivity (Force Range) Adjustment 6
    Creating a Baseline for the Potentiometer 7
  Conditioning the Sensors 7
Setting up the OEM Development Kit 8
  Viewing Measurements with the MicroView Display 8
  Using a Sensor with Two Pins 9
Using the MicroView Software 9
  Calibration 12
    Linear Fit Calibration 12
    Power Law Fit Calibration 14
  Tare 18
Using the Excel MicroView Reader 19
Modifying the Arduino Code 23
Support 23

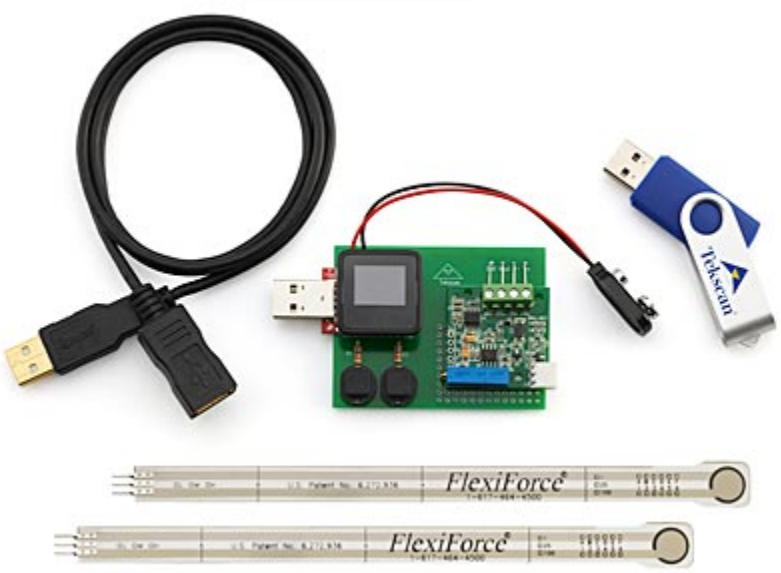
INTRODUCTION

The OEM Development Kit allows the user to view real-time pressure data using the Tekscan FlexiForce™ sensors.

OEM DEVELOPMENT KIT HARDWARE

Components List

The OEM Development Kit ships with the following components:

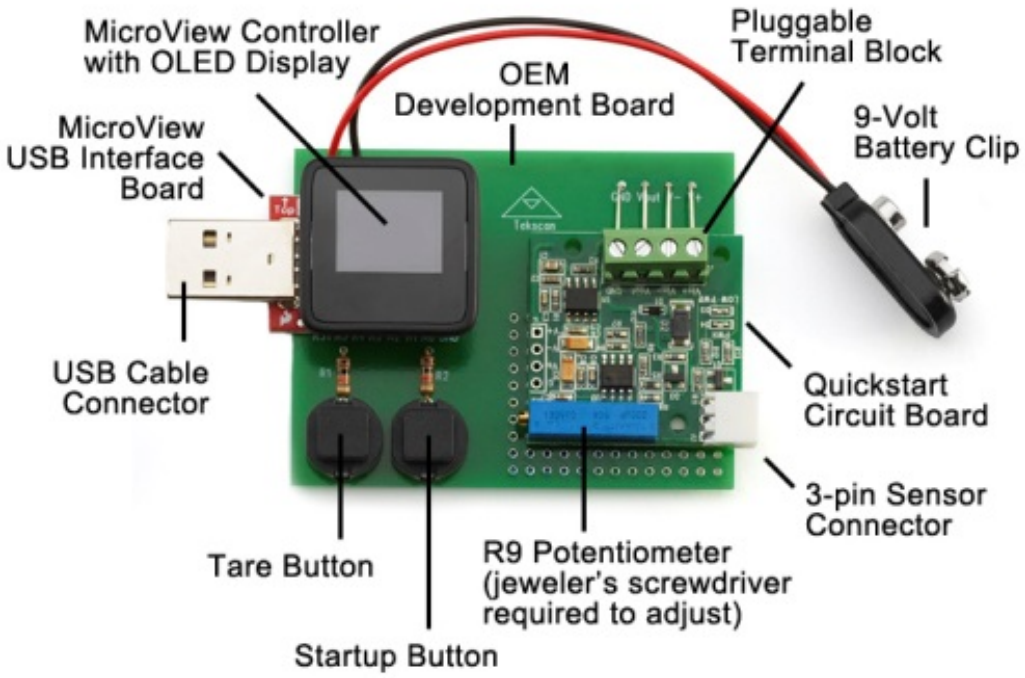


- (1) FlexiForce OEM Development Board
    - MicroView Display
    - Quickstart Board
    - Battery Clip
- (2) FlexiForce Sensors
- (1) Flash Drive with Software
    - FlexiForce MicroView (.exe file)
    - FlexiForce Micro 10 Hz (.ino file)
    - Flexi Micro Reader (.xlsm file)
- (1) USB Extension Cable
- 100kΩ Resistor

The FlexiForce™ OEM Development Board

The FlexiForce OEM Development Board is composed of a MicroView USB Interface Board and MicroView Controller with OLED Display, two Tactile Buttons, the Quickstart Circuit Board with Potentiometer, Pluggable Terminal Block, and 3-pin Sensor connector, and an attached 9-volt Battery clip. The Arduino Circuit Board converts the analog signal coming from the Quickstart Circuit Board into a Digital Signal, and outputs this signal on the MicroView Display (or on the included Microview software).

The following image shows the components that make up the OEM Development Board:



Specifications

- Input: 5.0V – 9V (for a constant power supply)
    - 9V battery recommended to keep a constant reference voltage.
- Output: 0.5V – 5V nominal (Vin ≥ 5V).
- Power: Green LED
- Low Power (Vin < 4.9V) = Yellow LED
    - Low power LED turns on when power source drops below 4.9V, as the supply voltage decreases, the maximum output voltage will decrease
- R9 (potentiometer): 15 turn, 500kΩ — adjusts gain and output range
- Vin- and GND are tied together in board

Signal Conditioning Circuit

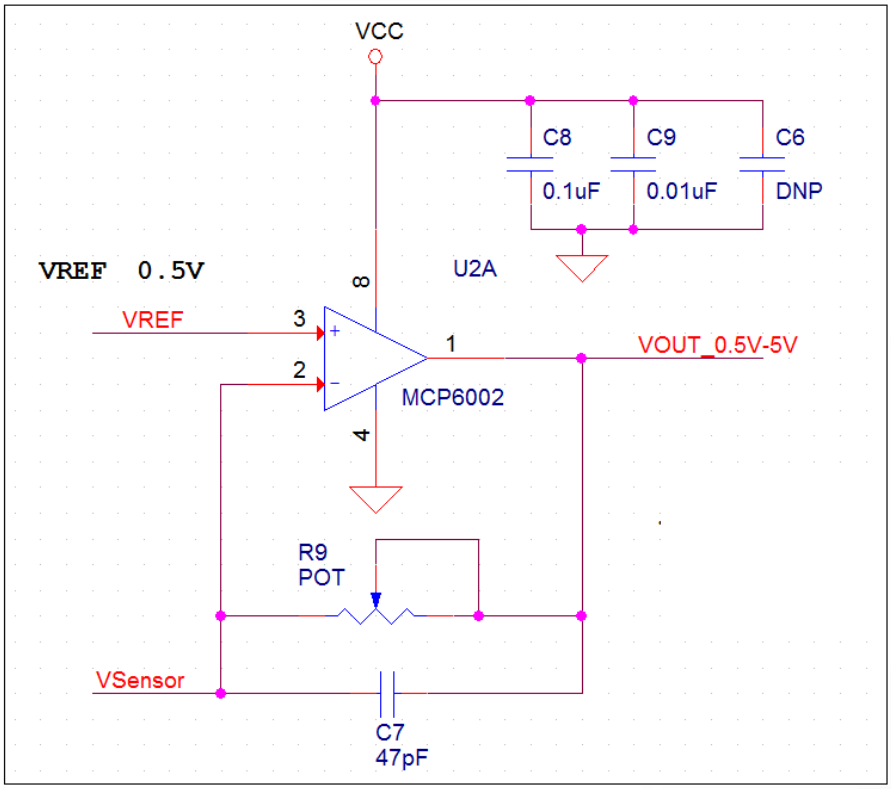


Figure 1 – Signal Conditioning Circuit

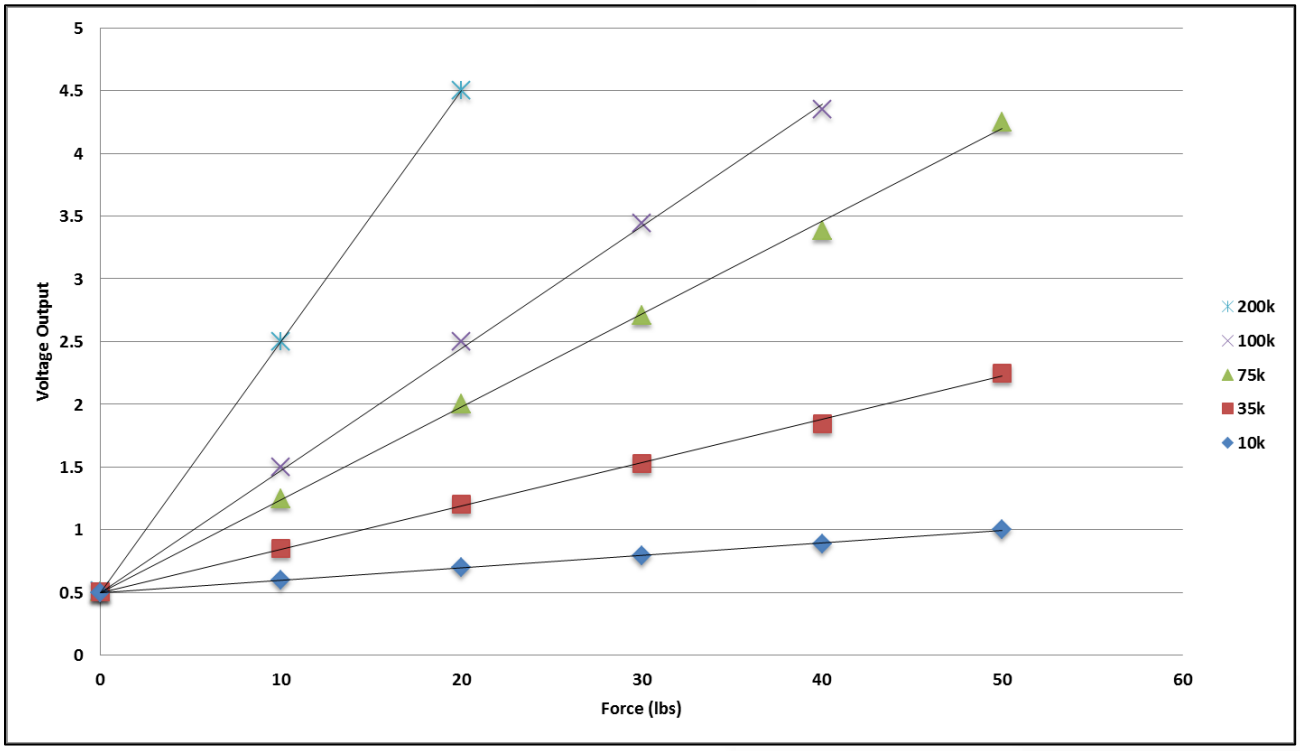
The MCP6002 op amps are specified at VDD – VSS 1.8V – 6V. The output equation is:

Vout = (1 + Rpot / Rsensor) * Vref

Sensitivity (Force Range) Adjustment

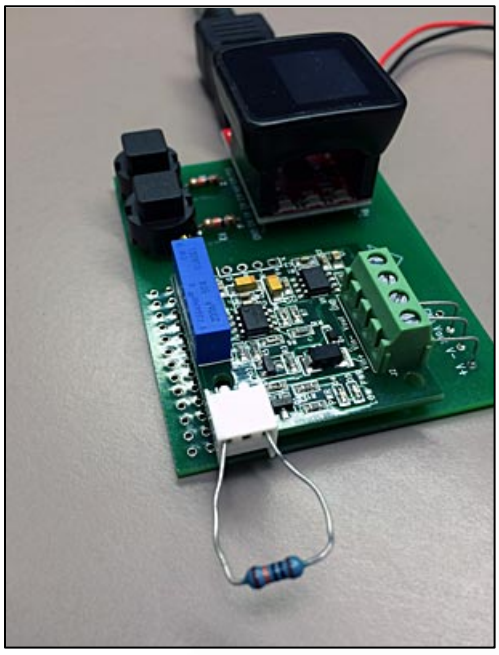
Changing the value of the feedback resistor via the Potentiometer adjusts the full scale of the force range. As the value of the feedback resistor increases, the maximum measurable force before the output saturates decreases. The opposite is also true, as the feedback resistor decreases the maximum measurable force that increases.

Note: A jeweler's screwdriver (not included) can be used to adjust the potentiometer.



The image above shows Voltage vs. Force: Varying Feedback Resistor with the Same Sensor.

Creating a Baseline for the Potentiometer



1. Power up the Development Board, as outlined in the Setting up the OEM Development Kit section.
2. Tare the output, as outlined in the Tare section.
3. Insert the included resistor into the two outer (the left and right, but not the middle) holes on the 3-pin sensor connector (shown at right).
4. Adjust R9 potentiometer with a jeweler's screwdriver, until the digital output reads 125 counts.

Conditioning the Sensors

For best results, we recommend conditioning the sensors before each use and before calibration. This process "breaks in" the sensor.

Place 110% (or more) of the maximum test load on the sensor for approximately 3 seconds. For example, if the maximum test load is 10 pounds, place 11 pounds onto the sensor. Remove the load from the sensor. Repeat 4-5 times. When finished, proceed to "Calibration."

SETTING UP THE OEM DEVELOPMENT KIT

Setting up the OEM Development Kit is quick and easy. Follow the procedure below.

1. Insert one of the FlexiForce Sensors face up into the white 3-pin connector slot on the Board.
2. Power up the system. You have two options:

   a) If you are planning to take measurements using the MicroView software on your computer, use the USB Extension Cable to power the Development Board. Insert one end into the Arduino Circuit Board's USB connector, and then insert the other end of the Cable into the USB port on your computer. The MicroView Display will illuminate with the Tekscan FlexiForce logo (shown at right).

   Note: the USB Cable must be connected in order for you to use the MicroView.exe software to read the data.

   b) If you plan to power the Development Board in a location without a computer, and view the measurements on the MicroView display only, you can connect a standard 9-volt battery (not included) to the Development Board's battery clip. This allows the system to be fully portable.

Viewing Measurements with the MicroView Display

Once the system is powered, either with the USB Cable or a 9-volt battery, you can view measurements on the MicroView display. Press the right tactile button (**Startup** button) on the Development Board (shown at right).

When you apply pressure to the sensor, the dial on the Display responds to this pressure (shown below).

Using a Sensor with Two Pins

When using the Quickstart Board with a two pin sensor, such as the A401 (not included) the center pin and the outer pin should be used as shown below.

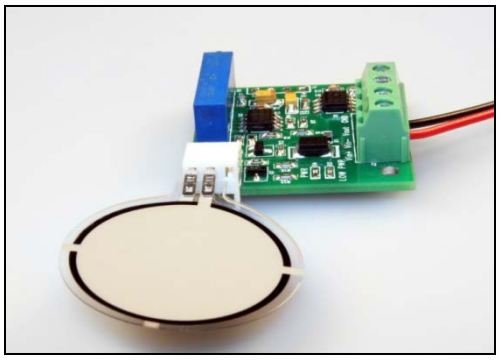


Figure 6 - Example of Using a Two Pin Sensor

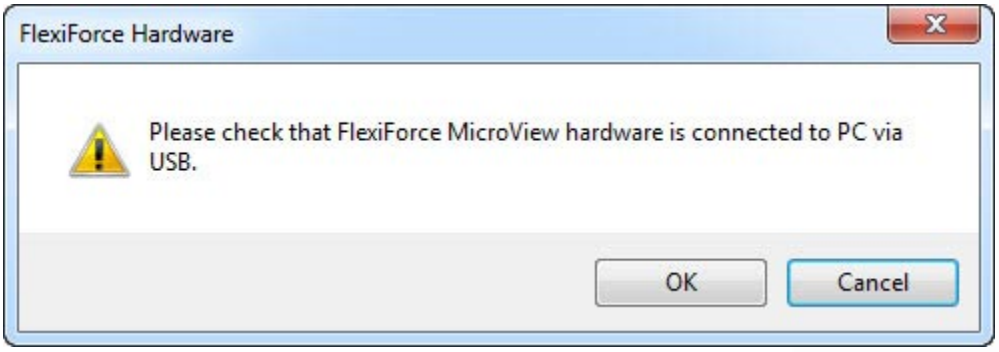
USING THE MICROVIEW SOFTWARE

To use the MicroView software, follow the procedure below.

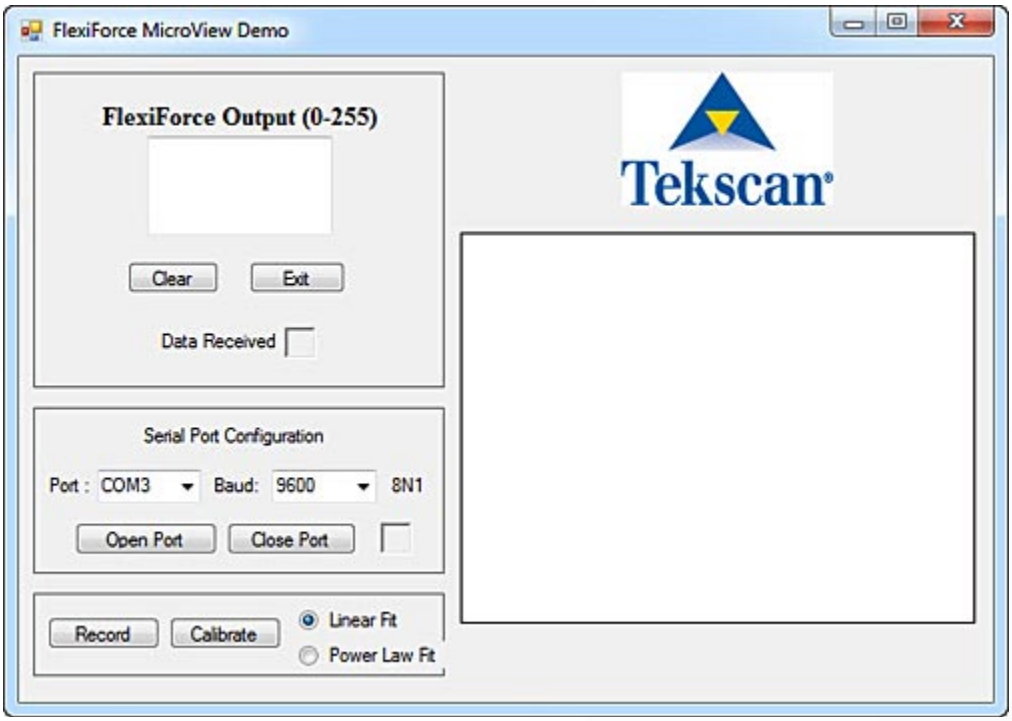
1. Set up the OEM Development Kit, as outlined above, via the USB Cable. When you see the Tekscan FlexiForce logo on the MicroView display, insert the Flash Drive into your computer's USB port. From your computer's Explorer window, locate and launch the **FlexiForce Microview.exe** software from the Flash Drive by double-clicking it.

   Note: You can copy the contents of the Flash Drive to a location on your computer and launch the software from your computer directly. In this way, you won't need the Flash Drive inserted in the computer's USB port each time you wish to use the software.

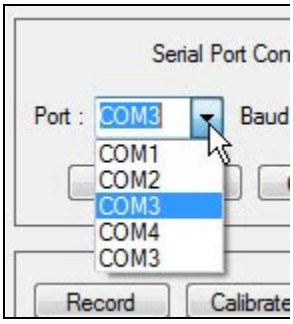
2. You may see the following dialog. Ensure your OEM Development Board is connected to the computer via the USB cable, and click **OK**.



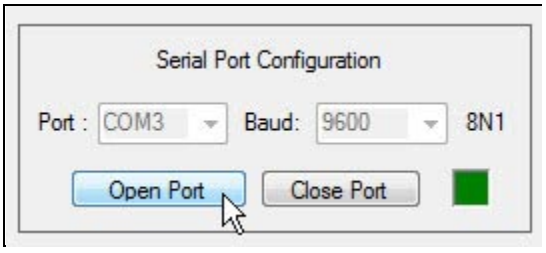
3. The FlexiForce MicroView Demo software opens on-screen (shown below).



4. Select the last **COM Port** from the "Port" drop-down list (in the example below, it is **COM3**).



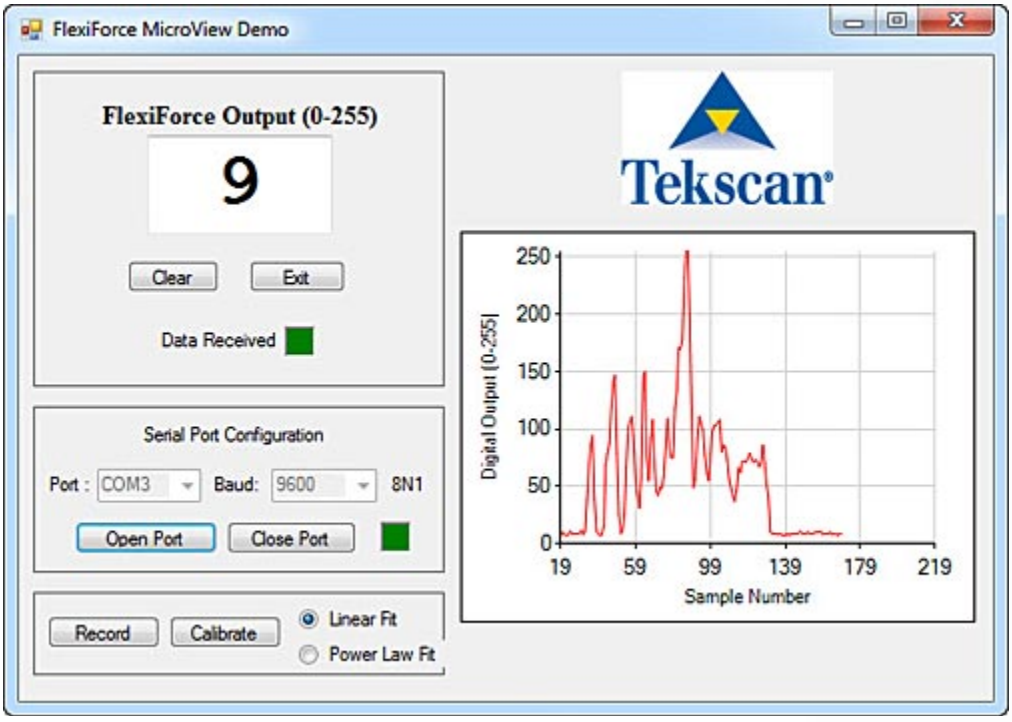
5. With the Port selected, click the **Open Port** button (shown below). You will see the color-square turn Green. This indicates the system is ready to display measurement data. If the square is red, this means you may not have the correct Port selected.



Note that the Port you should use for the MicroView hardware is usually the last one in this list. But if this Port doesn't work, try each Port sequentially, until the correct one is found.

6. With the correct Port selected, press the right tactile button (Startup button) on the Development Board (shown below).

7. Once started, apply pressure to the sensor. Both the MicroView display on the Board will display the pressure dial, and the software will display measurement data. At the top right of the screen, you will see a numerical readout in the FlexiForce Output window. At the left of the screen, you will see the graph display the Sample Number along the X-axis, and the Digital Output along the Y-axis (shown below).



Calibration

There are two Calibration procedures that can be used: **Linear Fit** and **Power Law Fit**. These two Calibration procedures are explained below.

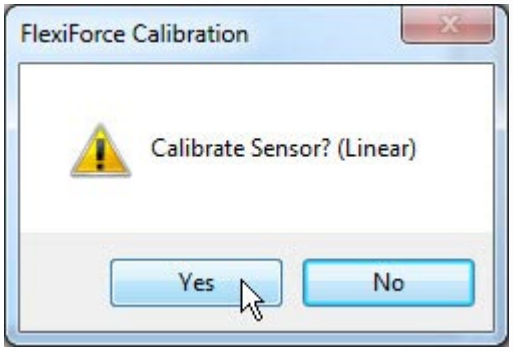
Linear Fit Calibration

Linear Fit. Linear calibration is a Single-Load Calibration, and assumes the sensor has zero output with zero applied load. The board will output approximately 0.5V at no load. The user applies a known force to get a single calibration point. The software then draws a straight line between the two points (zero point and the calibration point). The line is extrapolated to a digital output of 255.

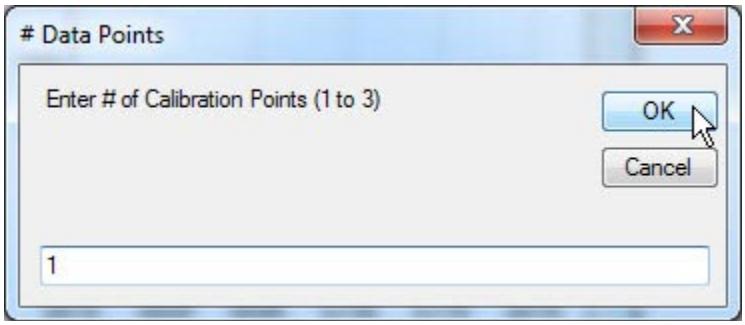
For experimental loads in the vicinity of the calibration load, single point calibration gives accurate results. If the experimental load is small or large compared to the calibration load, errors will grow. Typically, the graph of sensor Digital Output from varying load is a smooth curved line following a power law curve. A straight line from the origin is close to the curve near the calibration point, but increasingly divergent at the extremes.

Follow the procedure below to perform a Linear Calibration.

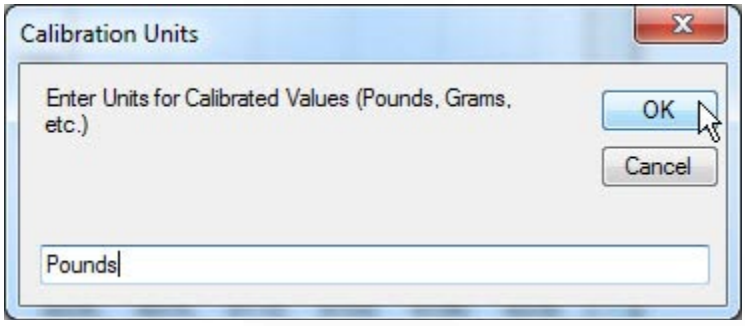
1. By default the **Linear Fit** radio button is selected. If it is not, ensure that it is selected, and then click the **Calibrate** button (shown below left). The Calibration dialog opens asking if you want to calibrate the Sensor. Click **Yes** (shown below right).



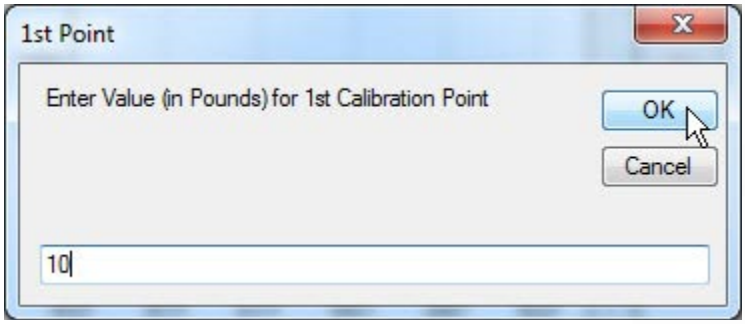
2. By default, 1 calibration point is selected. For most applications, this should be fine. However, you can select up to 3 Calibration points if you wish. When you've selected the number of Calibration points, click the **OK** button (shown below).



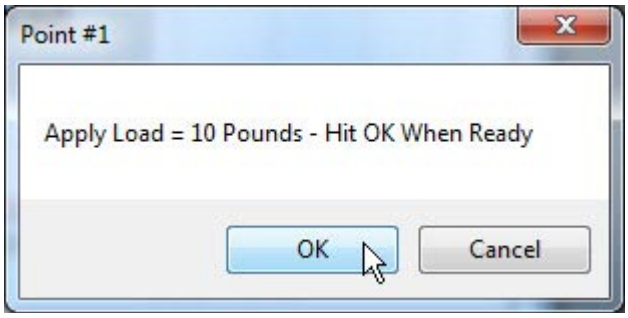
3. The next dialog asks you to specify the measurement units you are using. Enter this into the dialog and then click the **OK** button (shown below).

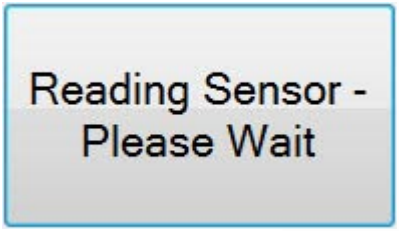


4. Enter the measurement Value in the next dialog. This is the number of units that the sensor will be measuring during the calibration process (shown below). After entering a value, click the **OK** button.

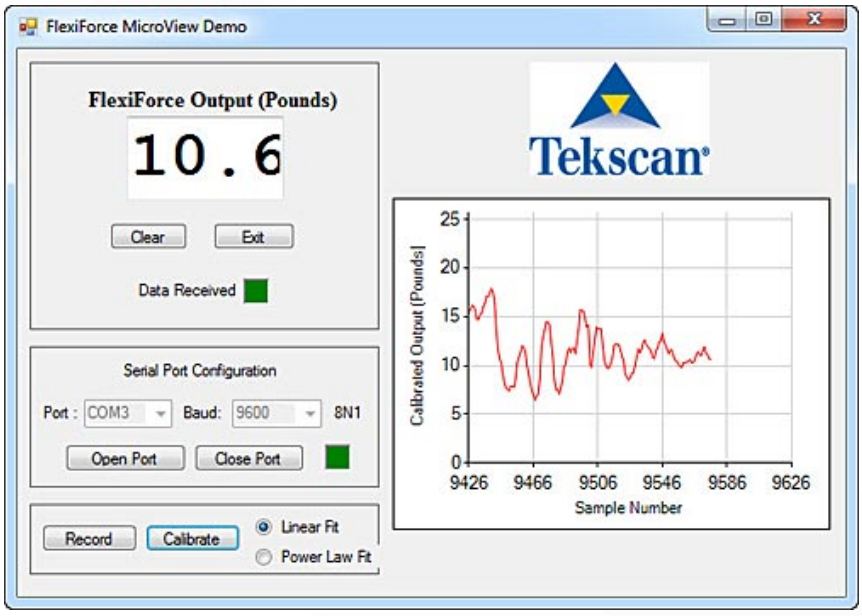


5. Apply the load to the sensor. In the example here, we are applying a 10 pound load to the sensor (shown below left). Next, click the **OK** button. The software takes a few seconds to Calibrate and displays the message shown below right.





6. When the Calibration process is completed, the dialog displays the new Calibrated Value (shown below left). Click the **OK** button. The Calibration process is now complete. You will now see output measured in Pounds within the FlexiForce MicroView software window (shown below right).



Power Law Fit Calibration

Power Law Fit: Power Law calibration is best used if you are using compliant (soft) material on top of the sensor, which affects load. Power Law calibration uses the load distribution information, algebra, and a numerical technique (iteration) to calculate the power law equation. For this reason, the applied loads must be precisely known and entered into the software correctly, and the calibration loads must generate different pressure distributions. Doing so will yield an accurate sensor calibration. In Power Law Calibration, multiple points are sampled to calibrate the sensor.

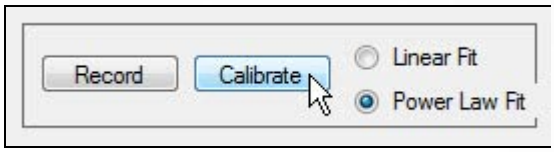
Power Law Calibration utilizes a sophisticated algorithm, which processes the distribution through a 'histogram' method. The two-point algorithm includes zero load to solve for the two constants in the exponential Power Law equation: $\mathbf{Y = AX^b}$

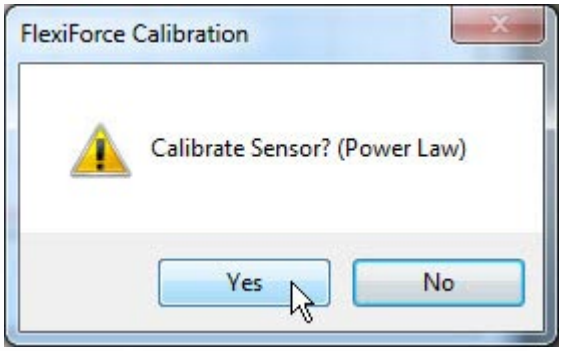
Y = force or load (in engineering units)
A = scale factor (determines slope)
X = raw digital output
b = exponent (determines curvature)

To perform a Power Law Fit calibration, the software requires 3 calibration points. We recommend setting Point 1 to 30% of the total experimental load, Point 2 to 60% of the total experimental load, and point 3 to 90% of the total experimental load.

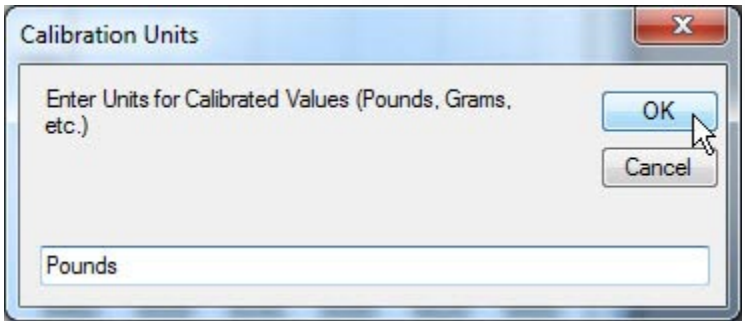
Follow the procedure below to perform a Linear Calibration.

1. Select the **Power Law Fit** radio button. Then click the Calibrate button (shown below left). The Calibration dialog opens asking if you want to calibrate the Sensor. Click **Yes** (shown below right).

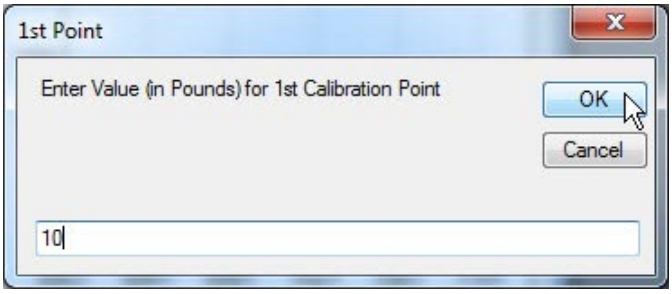




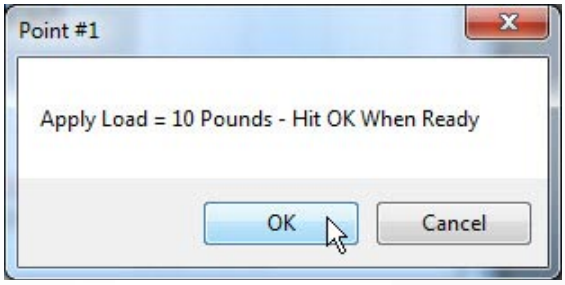
2. The next dialog asks you to specify the measurement units you are using. Enter this into the dialog and then click the **OK** button (shown below).

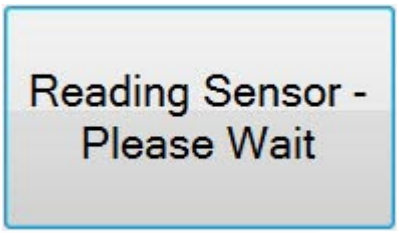


3. Enter the measurement Value in the next dialog. This is the number of units that the sensor will be measuring during the calibration process for the first point, which should be roughly 30% of the total test load. After entering a value, click the **OK** button (shown below).

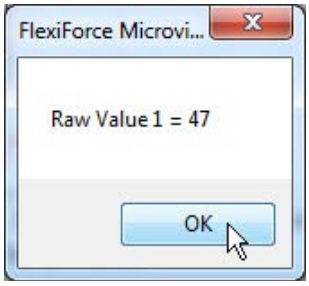


4. Apply the load to the sensor. In the example here, we are applying a 10 pound load to the sensor (shown below left). Next, click the **OK** button. The software takes a few seconds to Calibrate and displays the message shown below right.

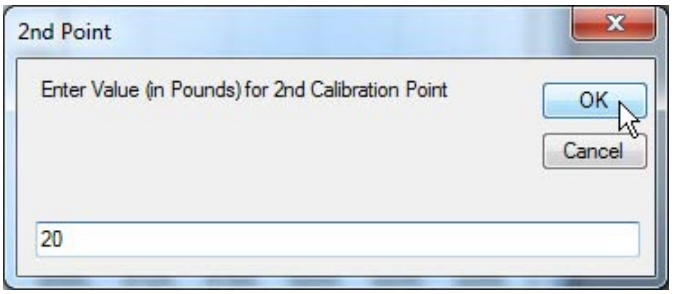




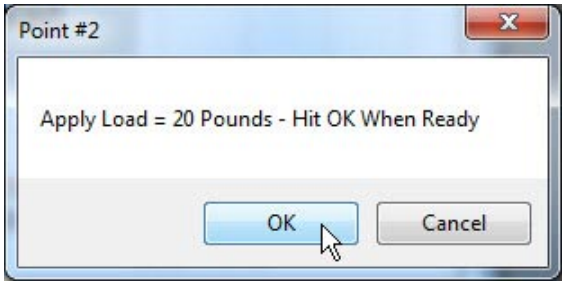
5. When the first point in the Calibration process is evaluated, the dialog displays the new Calibrated Value (shown below). Click the **OK** button.

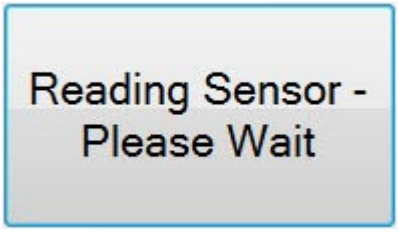


6. In the next dialog, enter the measurement Value for the second calibration point. This value should be roughly 60% of the total test load. After entering a value, click the **OK** button (shown below).

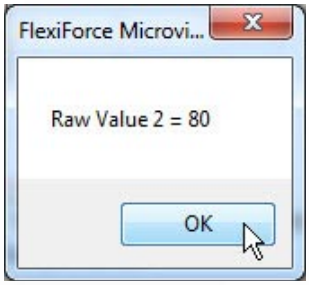


7. Apply the load to the sensor. In the example here, we are applying a 20 pound load to the sensor (shown below left). Next, click the **OK** button. The software takes a few seconds to Calibrate and displays the message shown below right.

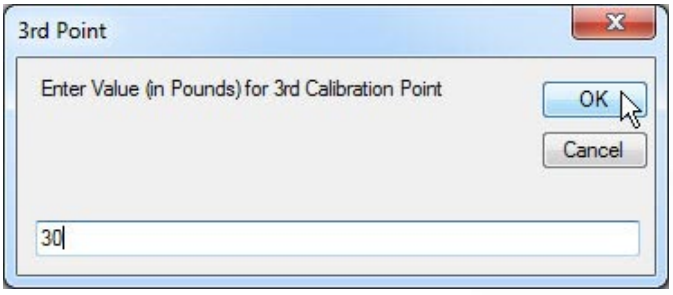




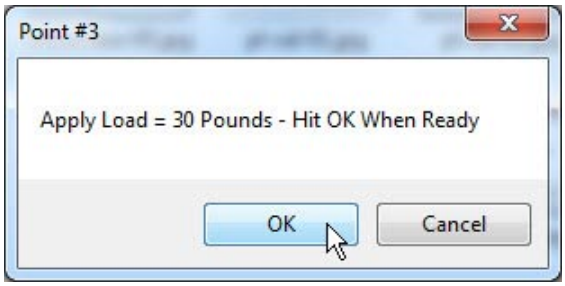
8. When the second point in the Calibration process is evaluated, the dialog displays the new Calibrated Value (shown below). Click the **OK** button.

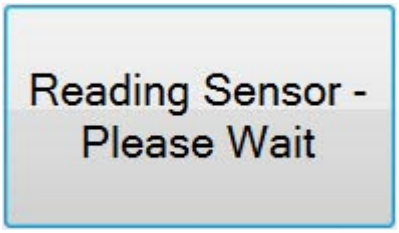


9. In the next dialog, enter the measurement Value for the second calibration point. This value should be roughly 90% of the total test load. After entering a value, click the **OK** button (shown below).

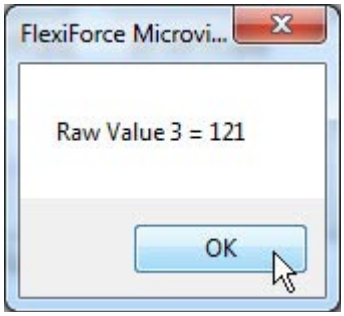


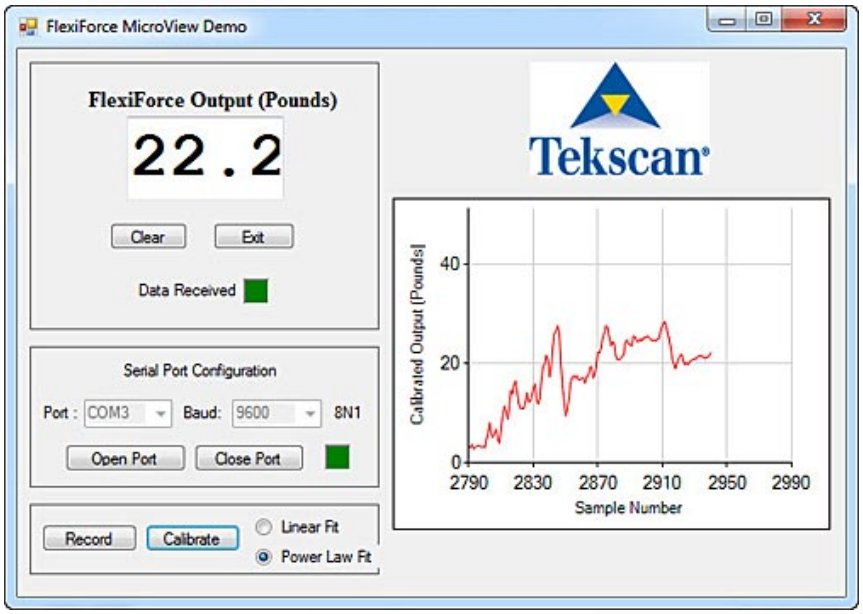
10. Apply the load to the sensor. In the example here, we are applying a 30 pound load to the sensor (shown below left). Next, click the **OK** button. The software takes a few seconds to Calibrate and displays the message shown below right.





11. When the third - and final - point in the Calibration process is evaluated, the dialog displays the new Calibrated Value (shown below left). Click the **OK** button. The Calibration process is now complete. You will now see output measured in Pounds within the FlexiForce MicroView software window (shown below right).





Tare

The digital output of the FlexiForce Sensor is 8-bit (256 counts). Tare allows you to reset the sensor to start counting from 0 at the load applied to the sensor. If there is no load on the sensor, and Tare is invoked, there is no effect on the Digital Output. If, for example, you place a load on the sensor and the load displays a count of 10, pressing the Tare button will offset the sensor count, and it will instead start counting upward from 0 only when a load of 10 or higher is applied. The sensor can still measure upward to 256 counts, but anything below a load of 10 will still measure 0 on the MicroView Display and within the software. Tare only affects the bottom end of the sensor and can be used to remove unwanted noise at extremely low pressures.

Since the Arduino Board is 10-bit (1024 counts) and the FlexiForce Sensor's Digital Output is 8-bit (256 counts), the Force Range of the FlexiForce Sensor (0-256) can be shifted anywhere within the 10-bit domain (0-1024). The Tare procedure allows you to shift the force range in this manner.

To use the Tare feature, following the procedure below.

1. Power up the MicroView Display with a 9-volt battery or USB cable. Press the right tactile button (**Startup** button) on the Development Board (shown at right).

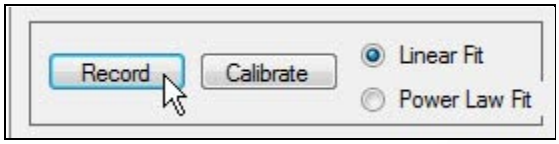
2. When you apply pressure to the sensor, the dial on the Display responds to this pressure (shown below).

3. Load the sensor with the pressures that you want removed from your measurement (your offset load). Press the left tactile button (**Tare** button). The Display shows that the software is "Taring" (shown below). You do not need to wait to release your finger from the Tare button. The software will Tare with a momentary click of the button.

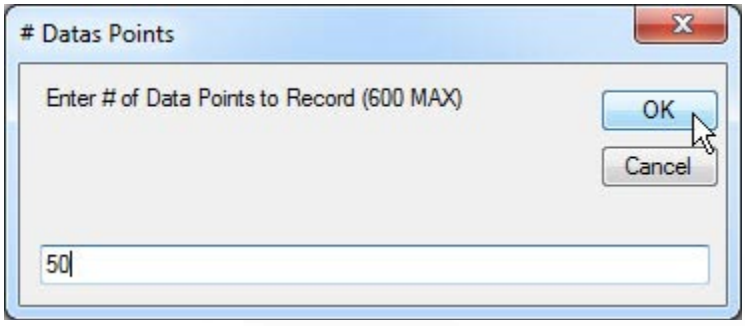
USING THE EXCEL MICROVIEW READER

You can import the data provided by the OEM Development Board into Microsoft Excel using the **FlexiMicroReader.xlsm** (macro-enabled worksheet). To do this, open the MicroView software, as outlined in the Using the MicroView Software section. Then do the following:

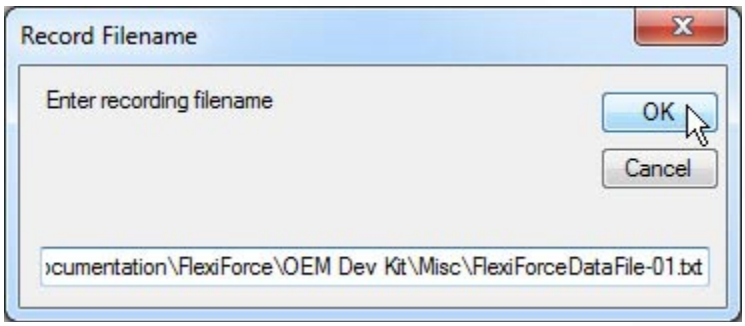
1. Click the **Record** button (shown below).



2. On the next dialog, enter the number of Data Points (up to 600) that you want recorded (shown below). Then click the **OK** button.

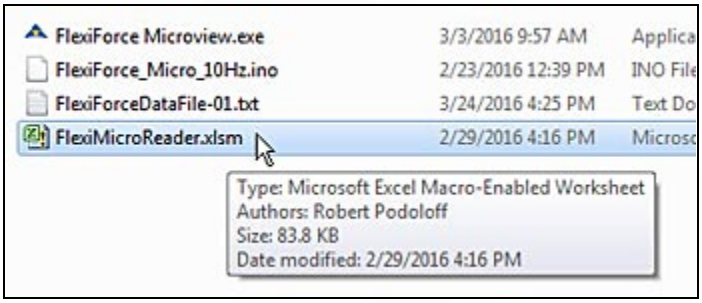


3. On the next dialog, enter the filename (along with the full path to the location of the File on your computer). Note: For easy retrieval later, it is recommended you place the file in the same folder as the **FlexiMicroReader.xlsm** file (shown below). When done, click the **OK** button.

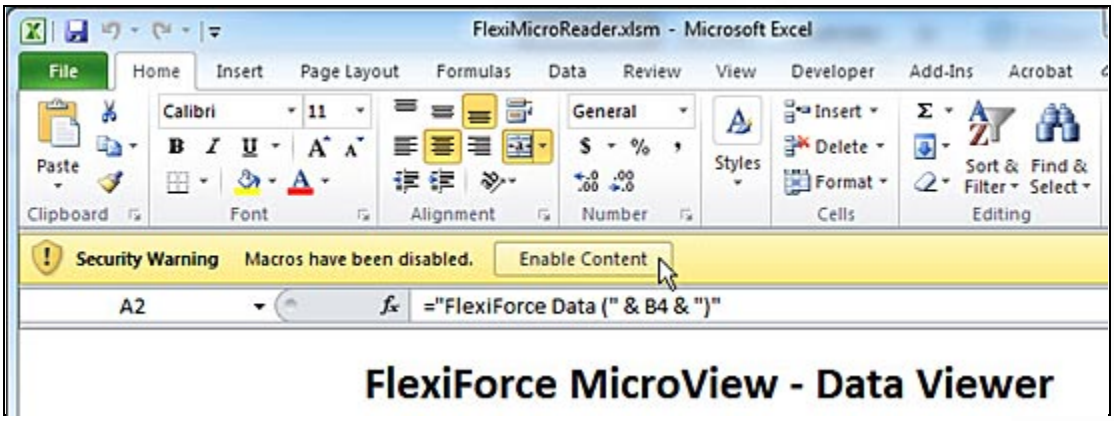


4. The next dialog puts the software in hold mode. Nothing will be recorded until you press the **OK** button to initiate the recording. Once this button is pressed, the recording immediately starts. When ready, press the **OK** button.

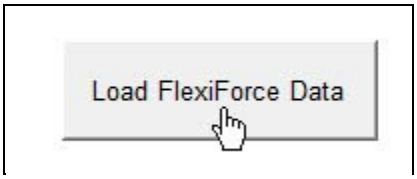
5. Once the number of points have been sampled, go into your Windows Explorer and locate the folder with both the **FlexiMicroReader.xlsm** and the data file you just recorded (in the example below, this is the **FlexiForceDataFile-01.txt** file). Double-click on the **FlexiMicroReader.xlsm** file (shown below).



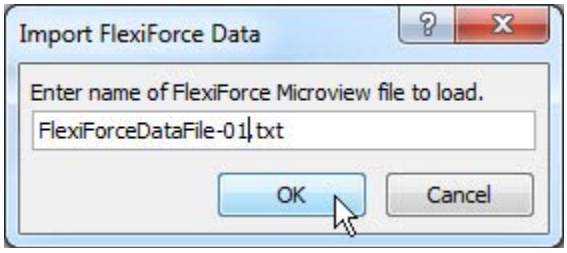
6. Microsoft Excel opens. Since this file contains Macros, you may have to first enable them by clicking on the **Enable Content** button (shown below). Do this now.



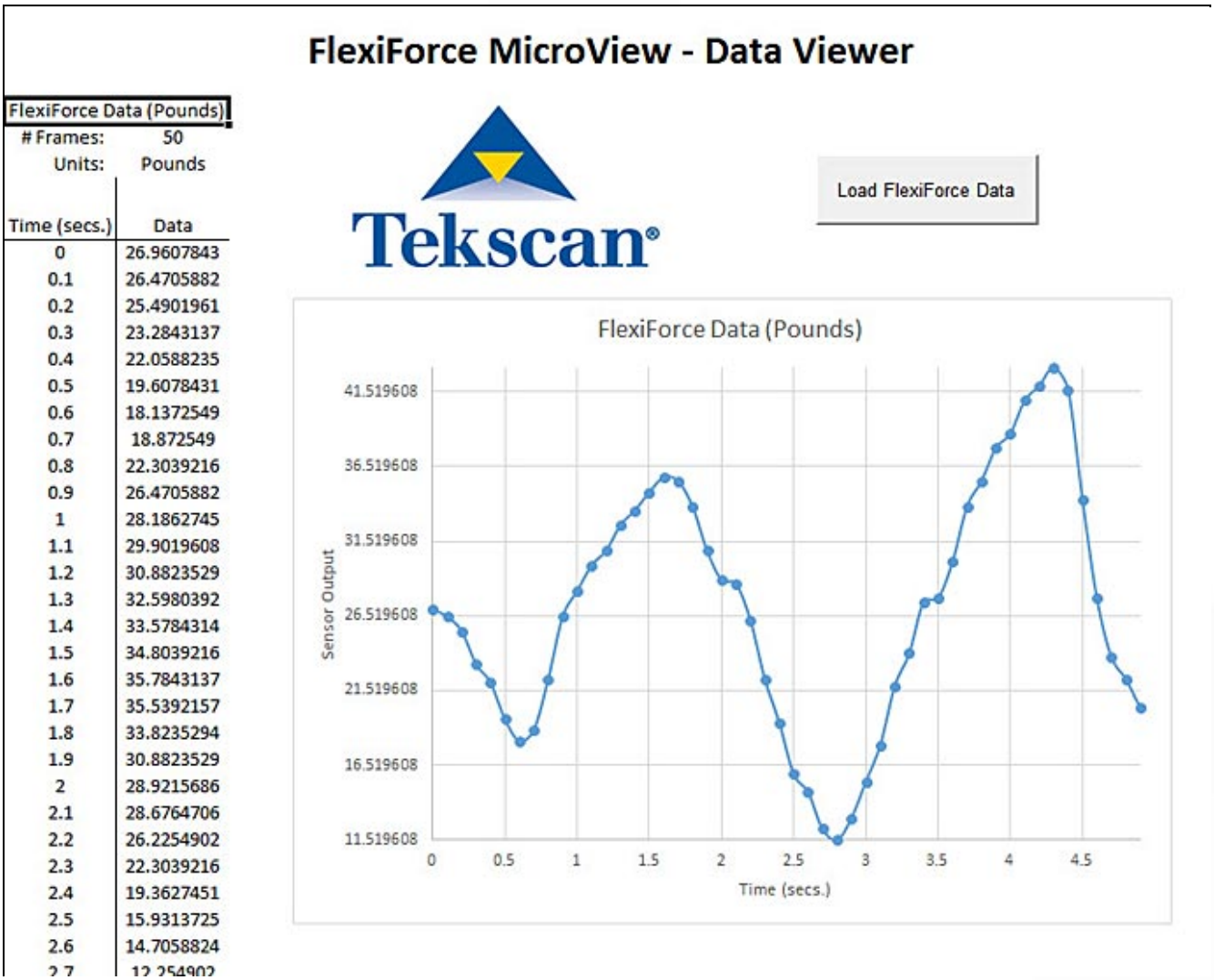
7. Next, click the **Load FlexiForce Data** button (along the right side of the screen – shown below).



8. The “Import” dialog opens. Since the data file is in the same location as the **FlexiMicroReader.xlsm** file, you do not need to enter the full path to find the file. Instead, simply enter the filename for the data file (in the example below, **FlexiForceDataFile-01.txt**). Then click the **OK** button (shown below).



The data from the file is loaded into the worksheet. The numerical data is displayed on the left side of the screen, and the data is plotted on the graph on the right side of the screen (shown below).



Note: You can load another file at any time by clicking the Load FlexiForce Data, entering a new file name, and clicking OK. The data from the new file overwrites the data from the old file.

MODIFYING THE ARDUINO CODE

The OEM Development Board uses the Arduino code, and as such, it is open source. If you wish to modify the Arduino code and reprogram it to your specifications, you will need to download the programming environment from Arduino. Follow the instructions below:

1. Go to https://www.arduino.cc/en/Main/Software
2. Click on “Windows Installer” and download the latest version from the Arduino download page.
3. Run “arduino-x.x.x.-windows.exe” from your download directory, where “x.x.x.” is the version you downloaded.
4. Install, using the default settings.

SUPPORT

Write, call, or fax us with any concerns or questions. Our knowledgeable support staff is happy to help you. Comments and suggestions are always welcome.

FlexiForce
a division of Tekscan, Inc.
307 West First Street
South Boston, MA 02127-1309

Phone: (617) 464 – 4500
Fax: (617) 464 – 4266
E-mail: flexiforce@tekscan.com