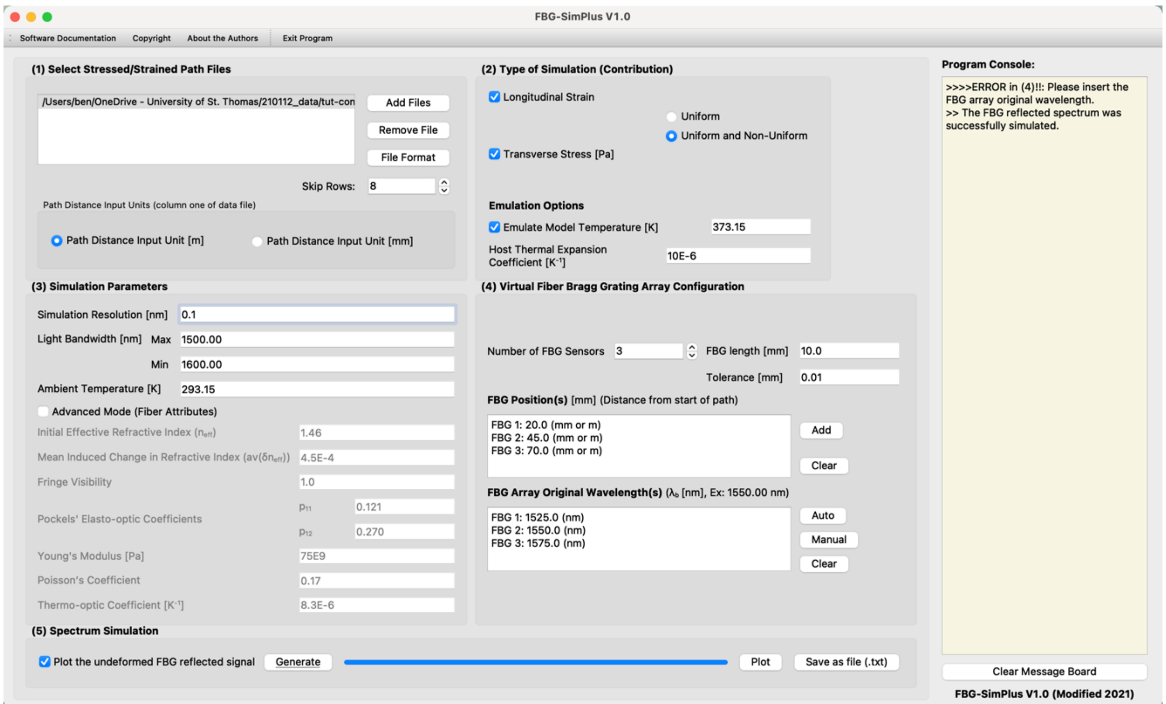
FBG-SimPlus V1.0

User Manual
January 2021



Improved Software By:

Benjamin N. Frey
Mentors: Ali Passian, Patrick Snyder

University of St. Thomas (St. Paul, Minnesota)
University of Illinois Urbana-Champaign
Oak Ridge National Laboratory

January 2021

Contact: freynben@gmail.com

Original Software By:

Gilmar F. Pereira (Technical University of Denmark)
Malcolm McGuga; Lars P. Mikkelsen

Table of Contents

Section 1: Info

1.1 Software Project Notice

First of all, I would like to thank Dr. Gilmar Pereira for his contribution to the original FBG_SiMul V1.0 software and opportunity of further expansion by making his project publicly available on GitHub (FBG_SiMul). Without the original foundation, our additional contributions to the software package would not have been possible. Parts of this user manual are adapted from his original work.

I would also like to thank my research mentors Ali Pasian (Oak Ridge National Laboratory), and Patrick Snyder (University of Illinois Urbana-Champaign) for their incredible patience, guidance, and motivation.

This research project was made possible by the generosity of the National Science Foundation and was primarily performed over the Summer of 2020 through the University of Illinois Urbana-Champaign's Physics Research Experience for Undergraduates.

This material is based upon work supported by the National Science Foundation and Department of Defense under Grants PHY-1659598 and PHY-1950744. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation or Department of Defense.

1.2 Copyright

The original FBG_SiMul author (Dr. Pereira) strongly believes in:

- The freedom to use the software for any purpose,
- The freedom to change the software to suit your needs,
- The freedom to share the software with your friends and neighbors, and
- The freedom to share the changes you make

We support these statements and continue to honor Dr. Pereira's original initiatives in FBG-SimPlus as well.

1.3 About the Software

FBG-SimPlus is a software package that simulates the reflected signal from a Fiber Bragg Grating (FBG) array along a predefined path in a Finite Element Method (FEM) model. FBG-SimPlus V1.0 differs greatly from the original software, FBG_SiMul V1.0, in both the underlying program methods and the additional feature of temperature contribution simulation. Dr. Pereira devised a helpful graphical user interface (GUI) that allows for users to easily modify simulation parameters without the requirement of expert knowledge. Together with this user manual you can find a folder containing files that are used in the tutorial section. If you found this software useful to your work, please give some recognition to the authors and cite this software (and the original version) together with the article DOI.

Section 2 FBG-SimPlus Software

This software was developed to simulate the implementation of FBG sensors into different types of physical structures. With a properly formatted input file, users can observe the effects of longitudinal strain (both uniform and nonuniform cases), transverse stress, and temperature on the reflected spectrum of an array of FBG devices embedded within a host material. This software presents the simulation of virtual FBGs which may be placed along a physical path within a FEM model. While this process requires stress and strain information to be exported from a user-defined path of a pre-existing FEM model, this path in principle can be arbitrarily drawn from anywhere within the model where the user desires to simulate the implementation of a sensor.

Figure 1 below demonstrates the placement of these virtual FBGs along a user-defined path on the boundary of a deformed mechanical cantilever. Of course, the fiber is not actually embedded in the FEM model. The utility of FBG-SimPlus is to propose this placement and the FBG's subsequent operation in theory.

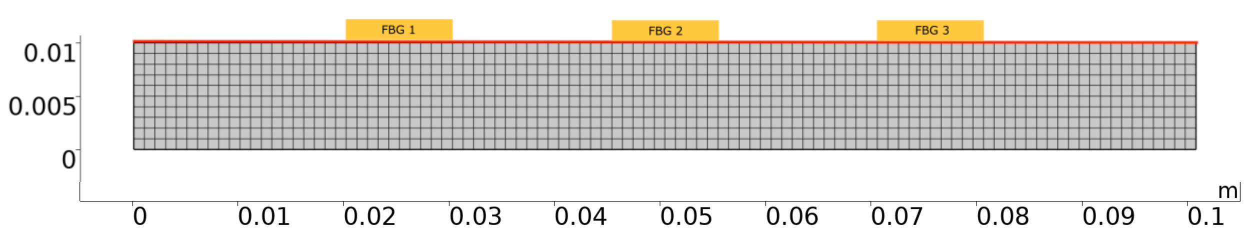


Figure 1: Virtual FBG path along cantilever. FBG dimensions not to scale.

2.1 Python Version

This software is provided in the form of a collection of Python files that can be easily modified. We recommend the use of a contained library environment such as Anaconda to install the Python programming language as well as the necessary modules. The required modules are required to run FBG-SimPlus V1.0:

- Python 3.8
- Modules: NumPy, PyQt5, Matplotlib, math, cmath

The software was developed with an object-oriented approach. The interface for the software is the widely available cross-platform Qt 5 framework for developing software with graphical user interfaces.

The following visual is adapted from the original user manual and illustrates the software's structure.

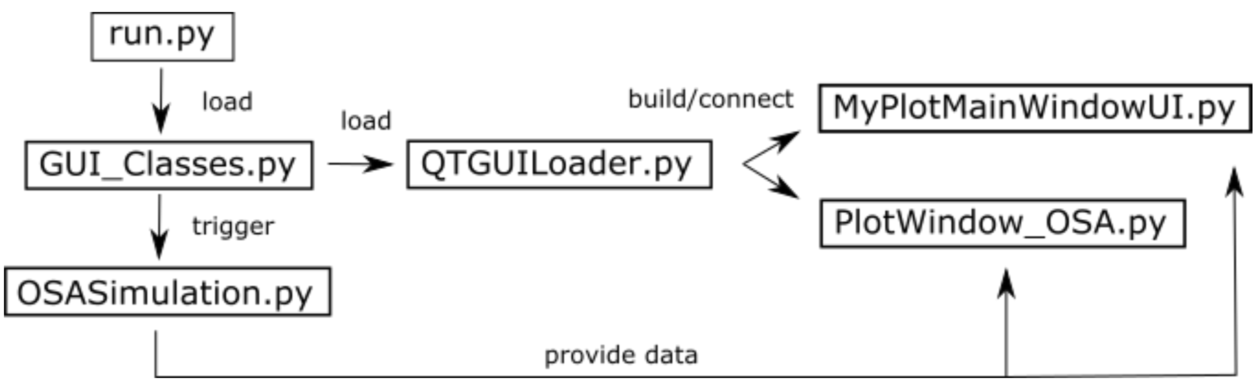


Figure 2: Revised program structure

2.1A Files/Structure

File: run.py
Description: To start the FBG-SimPlus code run the file "Run_me.py". This file will load the classes and packages needed to run the software.
Associated: Loads file "GUI_Classes.py"

File: GUI_Classes.py
Description: This class contains all the functions needed to load and start the user-interface. It checks if the input data files have the correct format and it triggers the FBG analysis classes.
Associated: Loads file "OSASimulation.py"
Important Functions:

init: Converts user-edited Qt Designer ".ui" files into a compiled Python format using the class "QTGUILoader". It connects all the actions in the user-interface with the underlying code in "OSASimulation.py"
actionOSAGenerate: Checks if all input data is formatted correctly. Calls the OSASimulation class, which simulated the FBG reflected spectrum

File: OSASimulation.py
Description: Class file to simulate the FBG reflected spectrum.
Important Functions:
UndeformedFBG: Simulated reflected spectrum for a non-deformed fiber.
DeformedFBG: Simulated reflected spectrum for a deformed fiber based upon input files and user-selected contribution types.

Section 3: FBG Spectrum Simulation

When running the program, the user should be greeted by the following screen seen in figure 3.

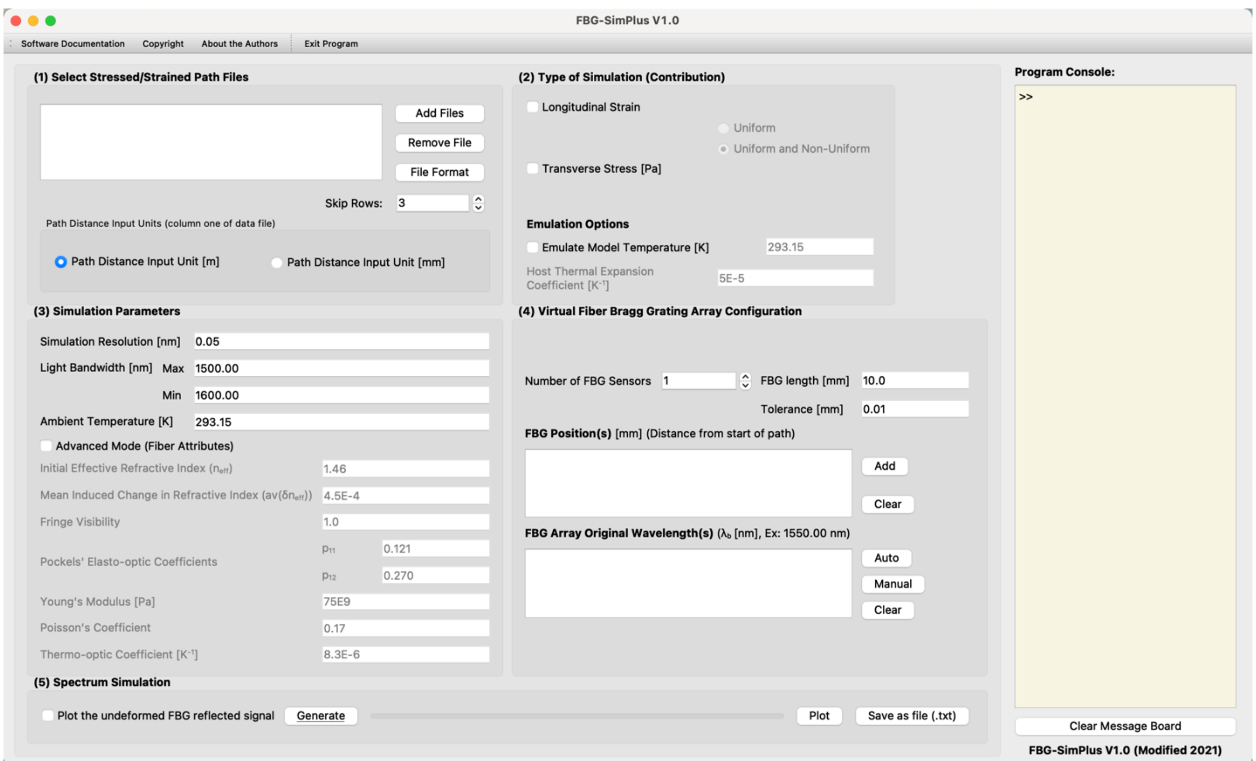


Figure 3: Initial user interface when FBG-SimPlus is launched

This window may be program into 5 primary sections:

1. **Select Stressed/Strained Path Files:** where users can import their properly formatted extracted data files
2. **Type of Simulation (Contribution):** allows the users to choose which components of the path deformation they would like to observe
3. **Simulation Parameters:** settings of the simulation that determine overall reflected spectrum simulation quality
4. **Virtual Fiber Bragg Grating Array Configuration:** allows the user to configure an array of FBGs to be placed virtually along the path
5. **Spectrum Simulation:** allows user to start simulation and view progress

A console window is also presented on the right side of the program. This console serves as a way to update users about the progress in conducting a simulation. The console can be cleared by clicking the button “Clear Message Board”.

3.1 Select Stressed/Strained Path Files

This section is where properly formatted data files may be read into the program. A properly formatted file is composed of 8 columns. Figure 4 below shows each of the following columns: 1.) Distance from start of path, 2.) The logarithmic true strain in the x direction (longitudinal direction of the fiber), 3.) The logarithmic true strain in the y direction, 4.) The logarithmic true strain in the z direction, 5.) The normal true stress in the x direction (longitudinal direction), 6.) The normal true stress in the y direction, 7.) The normal true stress in the z direction, 8.) The temperature.

0.0000	2.7538E-5	3.0171E-11	-6.8847E-6	7.1716E5	1.4343E5	-2.3283E-10	293.15
0.0010000	2.3320E-5	-5.2167E-6	-4.5259E-6	5.8013E5	-14392	-1.5134E-9	293.15
0.0020000	2.2533E-5	-4.4623E-6	-4.5178E-6	5.6357E5	1153.9	-1.6589E-9	293.15
0.0030000	2.2971E-5	-4.6374E-6	-4.5835E-6	5.7407E5	-1123.7	-5.8499E-9	293.15
0.0040000	2.2873E-5	-4.5644E-6	-4.5771E-6	5.7188E5	262.29	6.0099E-9	293.15
0.0050000	2.2719E-5	-4.5431E-6	-4.5438E-6	5.6796E5	8.6605	6.1846E-9	293.15
0.0060000	2.2523E-5	-4.5019E-6	-4.5052E-6	5.6309E5	58.567	-7.1595E-9	293.15
0.0070000	2.2303E-5	-4.4593E-6	-4.4606E-6	5.5756E5	15.293	-1.1147E-8	293.15
0.0080000	2.2075E-5	-4.4135E-6	-4.4151E-6	5.5187E5	19.070	-1.3926E-8	293.15
0.0090000	2.1839E-5	-4.3664E-6	-4.3678E-6	5.4597E5	10.589	-1.1350E-9	293.15
0.010000	2.1600E-5	-4.3187E-6	-4.3198E-6	5.3997E5	-0.32666	-3.1214E-8	293.15
0.011000	2.1362E-5	-4.2707E-6	-4.2722E-6	5.3402E5	4.4031	-7.5670E-10	293.15
0.012000	2.1122E-5	-4.2225E-6	-4.2242E-6	5.2802E5	3.4288	-6.9645E-8	293.15
0.013000	2.0881E-5	-4.1743E-6	-4.1759E-6	5.2199E5	-2.6634	-4.4791E-8	293.15
0.014000	2.0641E-5	-4.1260E-6	-4.1279E-6	5.1599E5	-2.7564	-1.1205E-8	293.15
0.015000	2.0403E-5	-4.0777E-6	-4.0801E-6	5.1001E5	2.7821	-7.9614E-8	293.15
0.016000	2.0162E-5	-4.0295E-6	-4.0319E-6	5.0399E5	-2.7882	-2.1886E-8	293.15
0.017000	1.9922E-5	-3.9812E-6	-3.9839E-6	4.9799E5	-2.7859	2.7387E-8	293.15
0.018000	1.9684E-5	-3.9328E-6	-3.9361E-6	4.9201E5	2.7712	-1.0598E-7	293.15

Figure 4: Raw FBG-SimPlus input data

The first column gives the distance from the start of the path in either millimeters or meters, depending on the units selected by the user. The interval between data points in this column is very important, as each point represents a discrete point along the path of the virtual fiber. In principle, it is possible to produce a more accurate simulation with smaller intervals between data points. This “distance” column must always align with the longitudinal direction of the virtual fiber. As we will see later, for studies where the path follows the shape of a deformed cantilever, this data column effectively becomes a parameterized path independent of the x-direction in our original coordinate system.

Columns two through four give the three normal components: x, y, z, respectively, of the true strain tensor. The true strain tensor is also known as the logarithmic strain tensor.

Columns five through seven give the three normal components: x, y, z, respectively, of the true stress tensor. The unit for each of these columns is either in Pa (assuming input unit “m” was selected), or MPa (assuming input unit “mm” was selected). Either way, our calculations are carried out in SI through doing unit conversion to SI within the program.

Column eight is the discrete temperature in Kelvin taken from each data point along the path. This column allows FBG-SimPlus to account for non-zero temperature gradients along the virtual fiber path. It is presumed that most imported datasets will have a uniform temperature distribution of 293.15 K. This is the ambient temperature (reference temperature) defined by our software.

The buttons “Add Files” and “Remove Files” facilitate which input files are visible to FBG-SimPlus. Upon clicking “Add Files”, the user should see a file directory display to choose a path to an input file. The button “File Format” allows users to view a brief reminder of mandatory input.

The incremental input box determines how many files of header information should be skipped within the input file. For example, in figure X above, only the first line would need to be skipped to reach the dataset. This example would correspond to an entry of “1” within the box.
The “Path Distance Input Units” selection area allows users to scale their datasets to SI units if needed. Most likely, the data will already be in SI units and the first option “Path Distance Input Unit [m]” should be selected.

3.2 Type of Simulation (Contribution)

This section allows users to consider the effects of particular contributions on the reflected spectrum of the fiber. It is important to note that none of these options change the information supplied from the input data file. By changing the individual parameters of this section, the user is merely experimenting with the individual effect of each of these real physical phenomena. **To obtain a simulation that most accurately reflects the supplied information of the input data file, it is recommended that the user select “Longitudinal Strain”, “Uniform and Non-Uniform”, and “Transverse Stress”.**

Longitudinal strain is characterized by an elongation (deformation) of a material. The effect that longitudinal strain has on the grating of a FBG is to either elongate the grating (resulting in a redshift in the reflected spectrum), or compress the grating (resulting in a blueshift in the reflected spectrum). The first option available to the user is to leave the "Longitudial Strain" box unchecked, resulting in FBG-SimPlus using the original grating period of the virtual FBG.

The second option available to the user is to select the "Longitudinal Strain" box as well as the "Uniform" box. This options results in FBG-SimPlus taking the mean of the values from column two of the input file within the FBG region. Observing figure X above, for a virtual FBG that starts at 0.05m and ends at 0.10m, this would be the mean of all the values between lines 6 through 11 from figure 4 as epsilon in equation X below. The third option available to the user is to select the "Longitudinal Strain" box as well as the "Uniform and Non-Uniform" box. This option results in FBG-SimPlus considering the individual strain values at each data point between lines 6 through 11 of figure 4 as epsilon. The virtual FBG region is effectively sliced into smaller segments, each with their own unique grating periods. Longitudinal strain by itself has the effect of shifting the central Bragg wavelength of the reflected spectrum. Non-uniform strain has the included effect of increasing the Full Width Half Maximum (FWHM) of the reflected waveform.

The inclusion of transverse stress results in the Birefringence effect or splitting of the waveform due to an induced difference in the directional refractive index of the material. FBG-SimPlus illustrates this effect to the user by producing two discrete waveforms in the reflected spectrum when the user selects "View Split Waveforms" in the plotting window. In reality, constructive summation of the waves would occur, and an observer would only notice one waveform with an increased FWHM. Under normal conditions, the effect of transverse stress on FWHM is much less than non-uniform strain. We encourage users to experiment with selecting the "Transverse Stress" box as well as leaving it unchecked to observe the contribution on reflected signal themselves.

An additional feature of FBG-SimPlus V1.0 is to allow the user the ability to model the effect of temperature on their existing datasets. This can be achieved by selecting the box "Emulate Model Temperature" and entering a temperature value in Kelvin into the box. Using this additional setting is not advised for models that have already been thermally manipulated (i.e. datasets that were not collected at a uniform temperature distribution). For datasets that were collected at the uniform ambient temperature, this option allows users to observe the additional effect of introducing the fiber to an elevated uniform temperature. The user defined "Host Thermal Expansion Coefficient" represents a_h in the equation 2 and should reflect the thermal expansion coefficient of the host material, the material that the FEM model was composed of.

$$P_e = \left(\frac{n^2}{2}\right)\left[P_{12} - \nu(P_{11} + P_{12})\right] \tag{1}$$

$$\Delta\lambda_b = 2n\Lambda\left[\{1 - P_e\}\epsilon + \left[\{1 - P_e\}\alpha_h + \frac{1}{n}\frac{dn}{dT}\right]\Delta T\right] \tag{2}$$

3.3 Simulation Parameters

The following parameters in Table 1 serve as important constants used in FBG-SimPlus that characterize the optical and mechanical properties of the virtual fiber.

Table 1: Default simulation parameters in FBG-SimPlus

Variable	Description	Example
Simulation Resolution	Spacing between discrete simulated wavelengths	0.05 nm
Light Bandwidth Min	Starting wavelength of graphed waveband	1500 nm
Light Bandwidth Max	Ending wavelength of graphed waveband	1600 nm
Ambient Temperature	Reference temperature for simulation	293.15 K
Initial Refractive Index	Effective refractive index of core and cladding	1.46
Mean Induced Changed in Refractive Index	Variation in refractive index along initial grating (difference between high and low layers)	4.5E-4
Fringe Visibility	Quantifies contrast of interference (usually 1)	1.0
Pockels' Normal Elasto-optic Coefficient	Normal component of elasto-optic tensor (p_{11})	0.121
Pockels' Shear Elasto-optic Coefficient	Shear component of elasto-optic tensor (p_{12})	0.270
Young's Modulus	Quantifies relationship between stress and strain	7.5 GPa
Poisson's Coefficient	Quantifies deformation perpendicular to load	0.17
Thermo-optic Coefficient	Change in refractive index with temperature	8.3E-6

By default, the "Advanced Mode (Fiber Attributes)" section is disabled, and the preceding values are used by default, representative of a silica glass fiber. To edit any of the preceding example values, check the box "Advanced Mode (Fiber Attributes)".

3.4 Virtual Fiber Bragg Grating Array Configuration

This section allows users to configure the virtual FBGs placed to be placed along the parameterized path. Because multiple FBGs can be chained together within a single optical fiber, it is possible for multiple FBGs to be placed at discrete locations along the path. These FBGs would typically have different initial Bragg wavelengths so that they may be distinguished from each other when simulating the reflected spectrum.

The "Number of FBG Sensors" allows the user to define the number of discrete FBG sensors that will be virtually placed along the path. At least one FBG is required.

The "FBG Length" controls the length of each FBG region. Typically, long period FBG sensors have a total length between 5mm and 10mm. This is the length of the FBG region defined where gratings exist within the fiber, not the entire length of the fiber itself.

The “Tolerance” parameter defines the distance tolerance as seen in column one of the input data file. With a tolerance of zero, the simulation will only consider elements that are fully contained within the defined FBG region. For example, for a virtual FBG that starts at 0.051m and ends at 0.101m with a tolerance of zero defined, this would infer that only the values between lines 6 through 10 from figure 4 would be considered as part of the FBG region, as the difference between 0.101m and 0.100m is greater than zero.

The “FBG Position(s)” attribute allows the user to define the position of each virtual FBG along the path. When pressing the Add button, the user is prompted to enter the distance from the start of the path at where the start of the FBG region position should be. Each FBG region is successively position in this fashion. For example, an FBG of length 10mm that is positioned a distance of 0.022m from the start of the path will terminate at 0.032m from the start of the same path. It is important the user positions the FBGs such that they do not overlap.

The “FBG Array Original Wavelength(s)” parameter defines the initial central Bragg wavelength for each FBG. It is beneficial for each FBG region to have its own distinct central wavelength to add adequate spacing between waveforms in the simulated reflected spectrum. By clicking the “Auto” button, the program automatically divides the entire waveband (defined previously) into n equal segments, where n is the number of FBG sensors. For example, for a waveband from 1500nm to 1600nm and three FBG sensors, each of the three FBG sensors will be assigned one of the following Bragg wavelengths: 1525nm, 1550nm, 1575nm. Typically, real FBG sensors are more aptly characterized by manufacturers through the initial grating periods instead of their directly related Bragg wavelengths. In the case of FBG-SimPlus, it is more efficient to define a wave range and ensure that the defined sensor Bragg wavelengths fall within this range. By clicking the “Manual” button, the user can enter their own custom Bragg wavelengths for each sensor.

3.5 Spectrum Simulation

This section allows the user to instruct the program to generate the reflected spectrum. By selecting the checkbox “Plot the undeformed FBG reflected signal”, each FBG is simulated as if no stress or strain is acting upon the fiber. This “undeformed spectrum” represents the initial unperturbed state of the fiber and serves as a reference. To view the simulated reflected spectrum due to stress and strain upon the fiber, the user must first click the “Generate” button. Here it is likely the user will be notified if any previously entered program parameters were entered incorrectly. Once the status bar has reached 100%, the user may click the “Plot” button to view a plot of reflectance as a function of wavelength.

The button “Save as file (.txt)” equivalently saves this generated graphical data into a text file for optional further post-processing by the user.

Section 4: Tutorial Cantilever Beam Study

4.1 COMSOL Multiphysics Model

Now we follow an example simulation using COMSOL Multiphysics to conduct a FEM analysis on a cantilever beam. The results of this study may be exported, formatted, and imported into FBG-SimPlus. Within FBG-SimPlus, the user may place an array of virtual FBGs along the exported data path from the model. The user can then simulate the reflected spectrum of this sensor array and graphically view the results.

4.1A Introduction to COMSOL Multiphysics

COMSOL Multiphysics is a cross-platform finite element analysis, solver, and multiphysics simulation software. The platform allows users the convenience of physics-based user interfaces while supplying the complexity of a coupled systems of partial differential equations solver. COMSOL is a modular program that allows users to customize their licenses by adding additional modules, each with a unique emphasis into some area of physics. One of the most commonly used modules in mechanical analysis is the “Structural Mechanics” module. This module is a collection of physics interfaces for COMSOL that includes: static, eigenfrequency, transiet, frequency response, parametric, transiet thermal stress, and other analyses for applications in structural mechanics, solid mechanics, and piezoelectricity. In addition to these interfaces, COMSOL offers various study options such as eigenfrequency, frequency domain, stationary, and time dependent. Each of these studies allow users to apply particular systems of equations to solve the current model at hand.

For our particular analysis, we have employed a “Solid Mechanics” interface with a “Stationary” study to observe the deformation of a 2-Dimensional cantilever due a single point load. We have also used the “Heat Transfer in Solids” interface as part of the Heat Transfer module. COMSOL allows users to define all of the parameters in regard to the geometry of their model as well as its material.

For this particular tutorial, we are using COMSOL Multiphysics version 5.4 with the Structural Mechanics and Heat Transfer modules.

4.1B Cantilever Beam Setup

First begin by opening COMSOL Multiphysics and selecting the option “Model Wizard” to assist in model creation. Then for “Space Dimension”, select “2D”. Much of the following tutorial also applies to 3-Dimension models, 2D was merely chosen for simplification. Under “Physics”, select “Structural Mechanics -> Solid Mechanics” and click “Add”, then “Study”. The “Study” selection should now be visible. Select “General Studies -> Stationary” and click “Done”. The main user interface should now be visible.

Next, it is time to set the model parameters under “Global Definitions”. Enter the following parameters seen in figure 5:

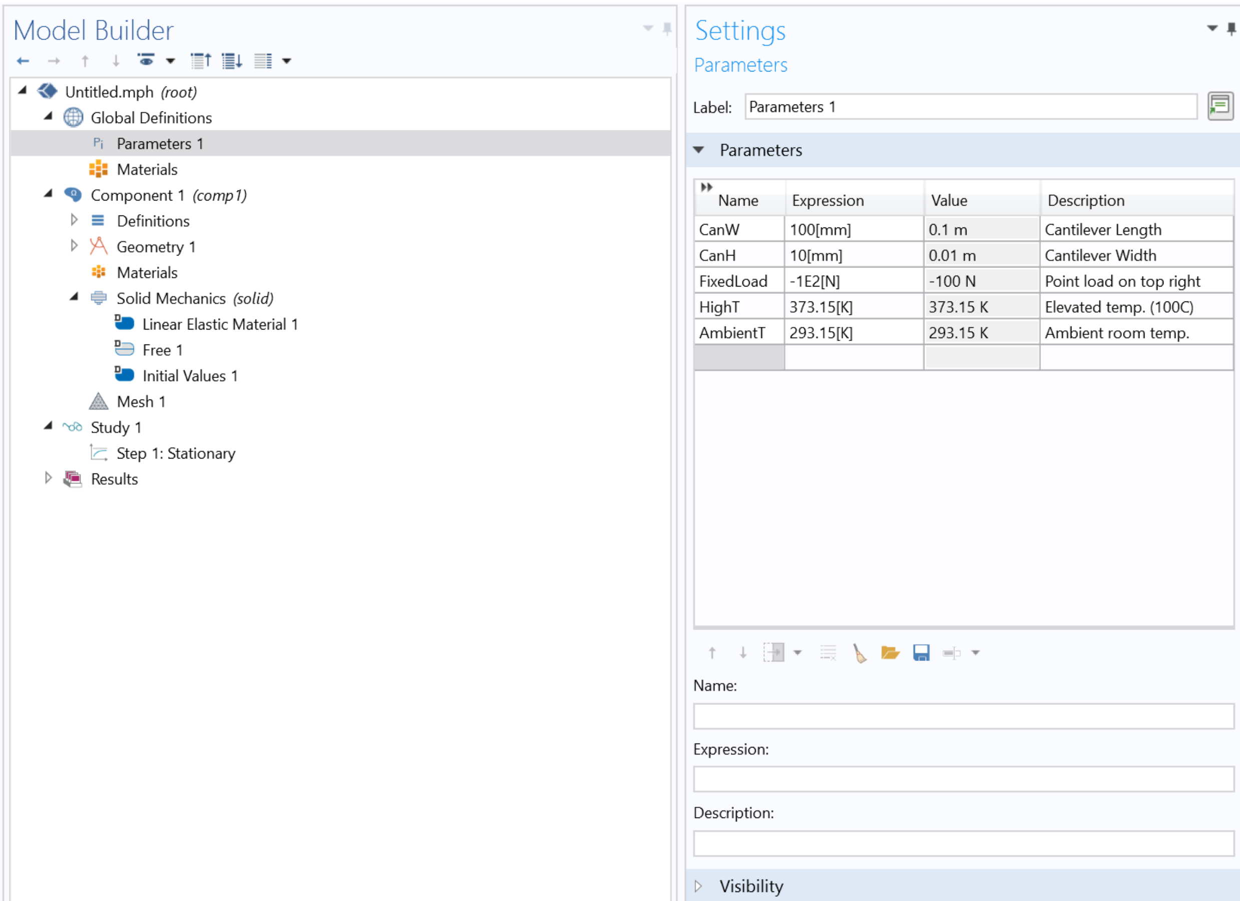


Figure 5: Cantilever parameter settings

After the parameters have been entered, select “Component 1 -> Geometry 1” and navigate to the “Geometry” tab of the menu. Select the rectangle tool and draw a rectangle in the “Graphics” tab. Now click on the grey rectangle. Under the settings window, set the width and height parameters to “CanW” and “CanH”, respectively. Set the x and y positions both to 0. Then click “Build All Objects”. You may need to zoom in to view the cantilever.

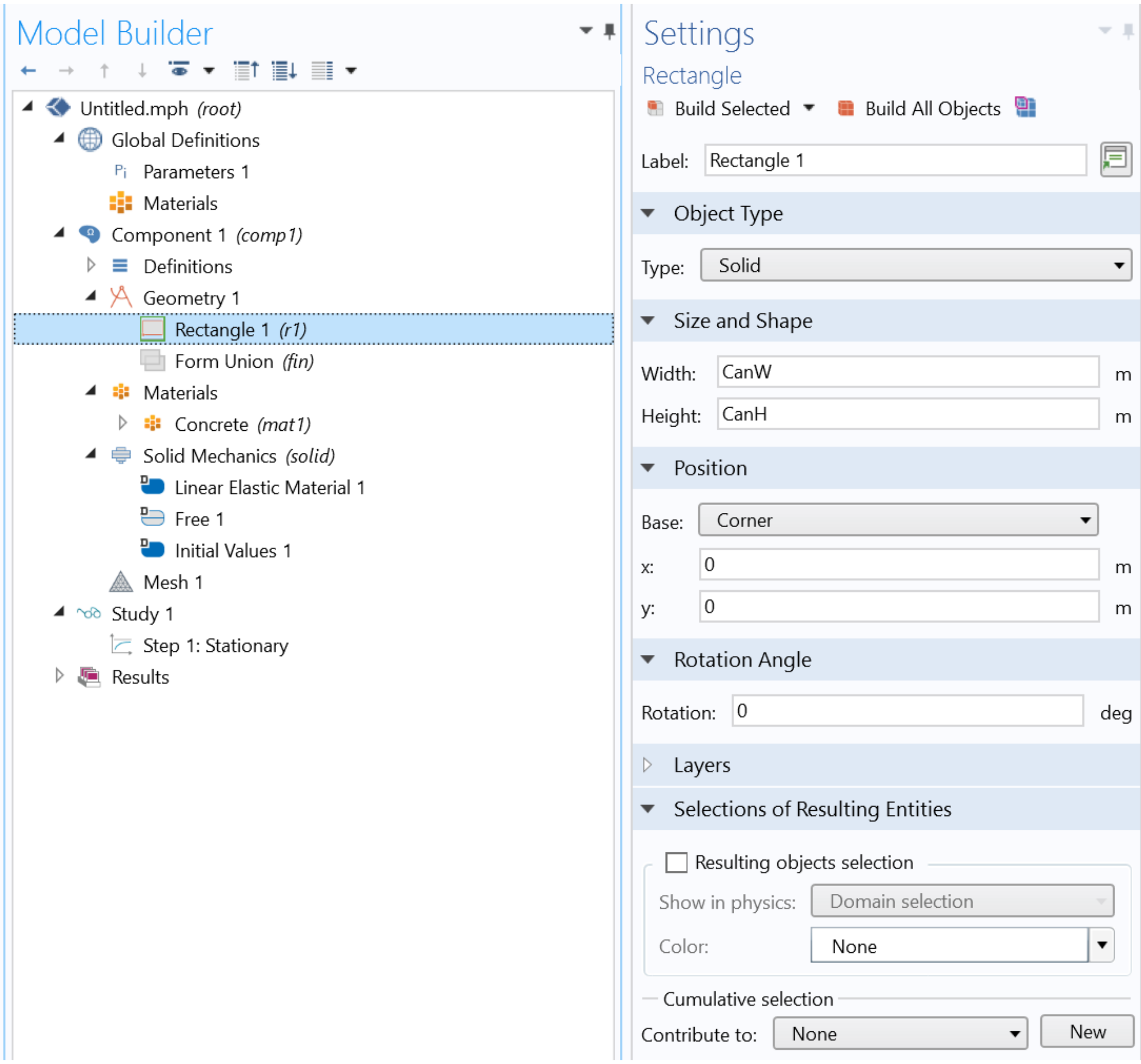


Figure 6: Rectangle geometry settings

Now it is time to select the material. Right-click on “Component 1 -> Materials” and select “Add Material from Library”. Then under “Built-in” select “Concrete”. Of course, this may be changed by the user later. You have now built a cantilever.

4.1C Study Setup

To manipulate the cantilever in COMSOL, the “Component 1 -> Solid Mechanics” and “Component 1 -> Mesh1” must be modified. First, click on “Component 1 -> Solid Mechanics”. Under the 2D Approximation, change “Plane strain” to “Plane stress”. This setting basically controls how COMSOL accounts for the missing 3rd dimension in our analysis and ensures that our study solutions give the correct strain values. Also change the “d” parameter under the “Thickness” section to “CanH”, which was the height of our cantilever. In essence, we are creating a 3D cantilever of height CanH, depth CanH, and longitudinal width of CanW.

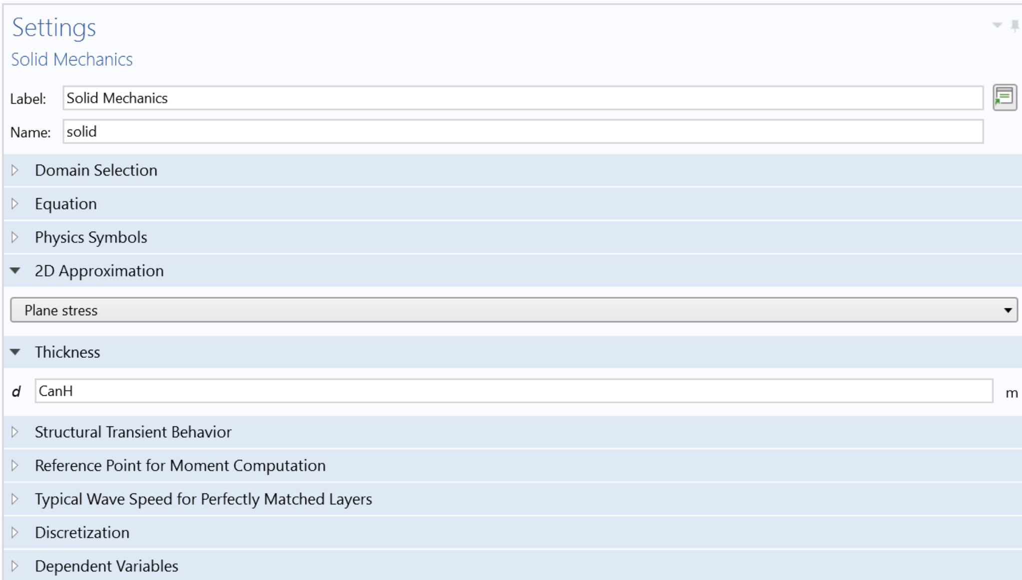


Figure 7: Solid Mechanics configuration

To add a constrain to the cantilever boundary, Right-click “Component 1 -> Solid Mechanics” and select “Fixed Constraint”. In the “Graphics” tab, select the vertical edge at x=0. This will be the fixed edge while the rest of the cantilever is free to deform. Now add the point load by right-clicking “Component 1 -> Solid Mechanics” and selecting “Points -> Point Load”. Within the “Graphics” tab, select the point at (0.10, 0.01), the top right corner. Under “Force” add “FixedLoad” for the y direction.

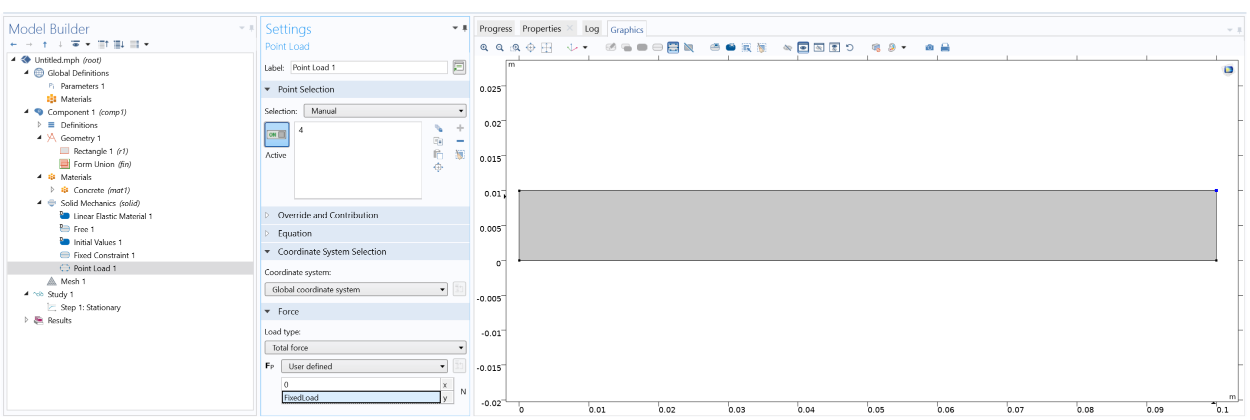


Figure 8: Point load configuration

Configuring the mesh in a certain way will be critical for exporting our data later from COMSOL. Because FBG-SimPlus prefers even intervals between data point in the first column of the input data, a custom mesh must be created for the cantilever. If we did not create a custom mesh, the model would be meshed automatically with triangular polygons to optimize computation time of the study. Our model is small enough that this optimization process won't be noticeably beneficial in computation time. First right-click "Component 1 -> Mesh 1" and select "Mapped". Then right-click on "Component 1 -> Mesh 1 -> Mapped 1" and select "Distribution". Again, right-click on "Component 1 -> Mesh 1 -> Mapped 1" and select "Distribution". There should now be a "Distribution 1" and a "Distribution 2" under "Component 1 -> Mesh 1 -> Mapped 1". Click on the first distribution and change the distribution type to "Explicit". Enter "range(0,1[mm],CanW)" as the value for relative placement of vertices along edge. Switch over the "Graphics" tab and click on the horizontal edge on y=0. Once clicking "Build All", you should see that the cantilever has been divided into 100 segments along the x-axis.

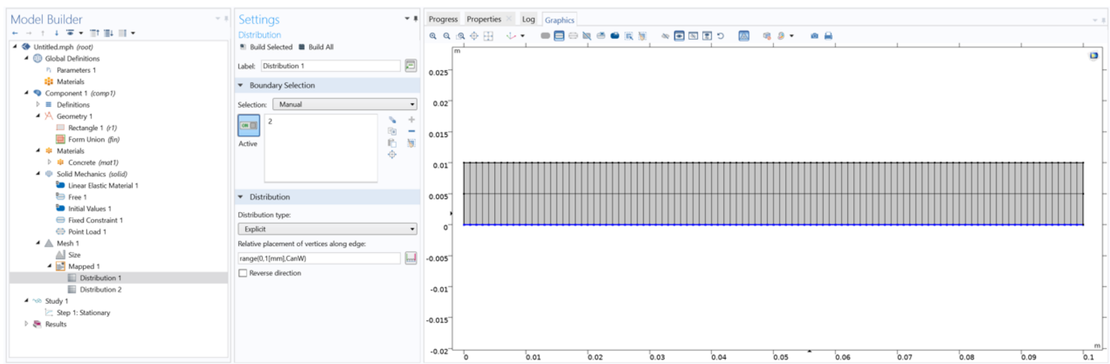


Figure 9: First mesh distribution

Click on the second distribution and change the distribution type to "Explicit". Enter "range(0,1[mm],CanH)" as the value for relative placement of vertices along edge. Switch over the "Graphics" tab and click on the vertical edge on x=0. Once clicking "Build All", you should see that the cantilever has been divided into 10 segments along the y-axis in addition to the mesh from the first distribution.

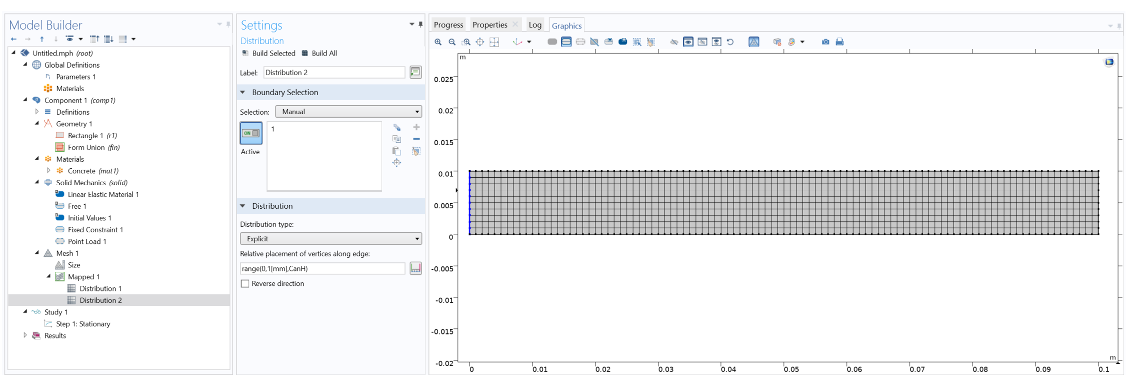


Figure 10: Second mesh distribution

To view the solution to this physics problem, click on “Study 1” then “Compute”. A colored plot of von Mises stress should pop up in the “Graphics” tab once the study has been solved.

Before extracting data, let’s go back and add a Heat Transfer physics to “Component 1”. Right-click “Component 1” and select “Add Physics”. Under the “Add Physics” tab, select “Heat Transfer -> Heat Transfer in Solids”. Right-click on “Component 1 -> Heat Transfer in Solids” and and select “Temperature”. Click on this newly added “Temperature” and change the temperature setting to “AmbientT”. This will ensure that our cantilever model is held at our pre-defined room temperature.

4.1D Extracting Data

Now that the cantilever model and study have been created, stress, strain, and temperature information need to be extracted from the model. It is possible to extract this information by defining a Cut Line 2D along the longitudinal direction of the beam. One particular path of interest can be seen in figure 1.

To utilize a Cut Line in COMSOL, right click “Results -> Data Sets” and select “Cut Line 2D”. Point 1 represents the start of the path. Based on our figure X above, we will want this point to be at (0, CanH). Point 2 is the end of our path which should be set at (CanW, CanH).

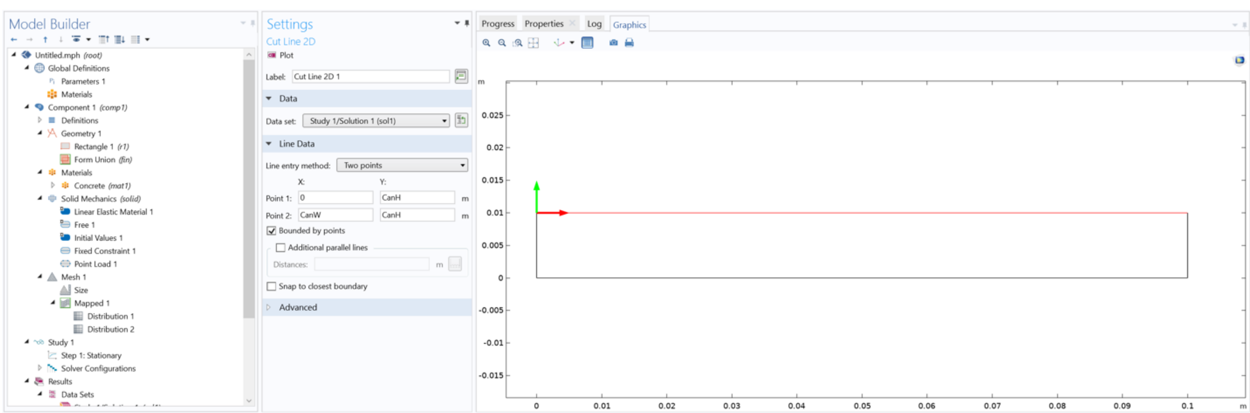


Figure 11: Cut line configuration on cantilever boundary

The nice thing about using a Cut Line is that when our cantilever beam is deformed, the Cut Line matches the new deformed boundary. This allows us to extract consistent path data from all types of deformations.

To export the study data, right-click “Results -> Export” and select “Data”. Select this newly added “Data1” and change the data set to “Cut Line 2D 1”. Add the following expressions:

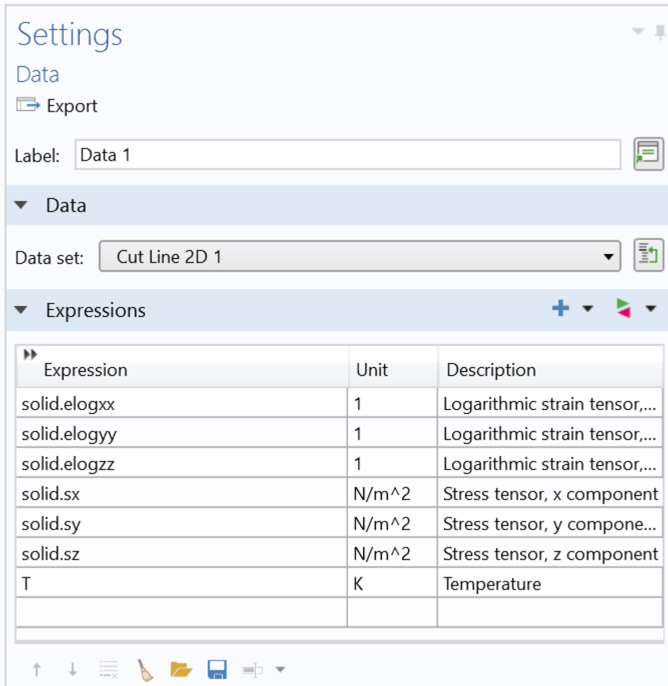
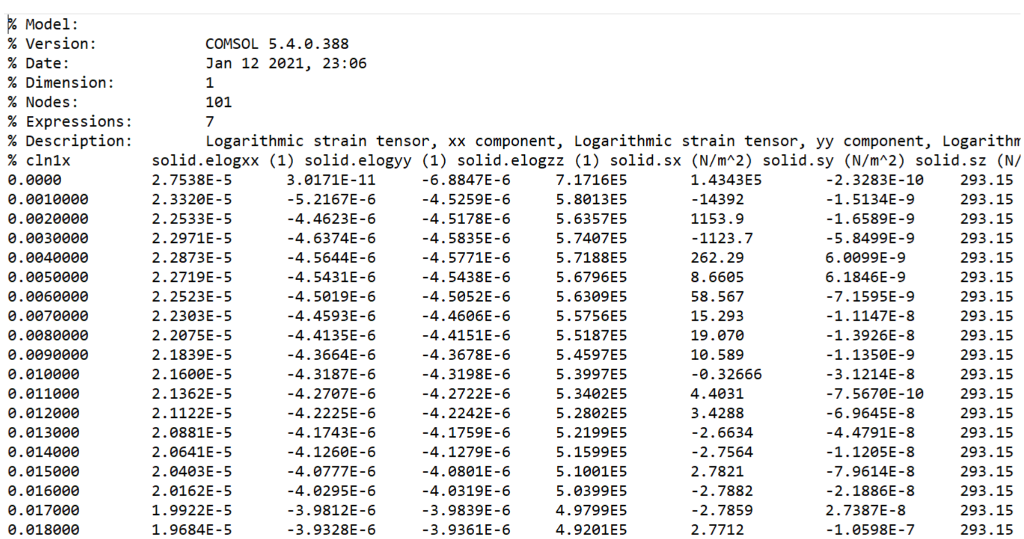


Figure 12: Export data settings to adhere to FBG-SimPlus format

Under the "Output" section, change space dimension to 1. This is because we are only concerned with the distance along the path in the longitudinal direction, we can delete all other distance-related data. Under the "Advanced" section, deselect "Full precision" and select "Sort". We only need a few digits of precision and this will help us sanitize our export data. Then click the "Export" button right underneath the Settings window. The exported file should like similar to this:



```
% Model:
% Version:          COMSOL 5.4.0.388
% Date:             Jan 12 2021, 23:06
% Dimension:        1
% Nodes:            101
% Expressions:      7
% Description:      Logarithmic strain tensor, xx component, Logarithmic strain tensor, yy component, Logarithm
% cln1x        solid.elogxx (1) solid.elogyy (1) solid.elogzz (1) solid.sx (N/m^2) solid.sy (N/m^2) solid.sz (N/
0.0000         2.7538E-5      3.0171E-11     -6.8847E-6     7.1716E5       1.4343E5       -2.3283E-10    293.15
0.0010000      2.3320E-5      -5.2167E-6     -4.5259E-6     5.8013E5       -14392         -1.5134E-9     293.15
0.0020000      2.2533E-5      -4.4623E-6     -4.5178E-6     5.6357E5       1153.9         -1.6589E-9     293.15
0.0030000      2.2971E-5      -4.6374E-6     -4.5835E-6     5.7407E5       -1123.7        -5.8499E-9     293.15
0.0040000      2.2873E-5      -4.5644E-6     -4.5771E-6     5.7188E5       262.29         6.0099E-9      293.15
0.0050000      2.2719E-5      -4.5431E-6     -4.5438E-6     5.6796E5       8.6605         6.1846E-9      293.15
0.0060000      2.2523E-5      -4.5019E-6     -4.5052E-6     5.6309E5       58.567         -7.1595E-9     293.15
0.0070000      2.2303E-5      -4.4593E-6     -4.4606E-6     5.5756E5       15.293         -1.1147E-8     293.15
0.0080000      2.2075E-5      -4.4135E-6     -4.4151E-6     5.5187E5       19.070         -1.3926E-8     293.15
0.0090000      2.1839E-5      -4.3664E-6     -4.3678E-6     5.4597E5       10.589         -1.1350E-9     293.15
0.010000       2.1600E-5      -4.3187E-6     -4.3198E-6     5.3997E5       -0.32666       -3.1214E-8     293.15
0.011000       2.1362E-5      -4.2707E-6     -4.2722E-6     5.3402E5       4.4031         -7.5670E-10    293.15
0.012000       2.1122E-5      -4.2225E-6     -4.2242E-6     5.2802E5       3.4288         -6.9645E-8     293.15
0.013000       2.0881E-5      -4.1743E-6     -4.1759E-6     5.2199E5       -2.6634        -4.4791E-8     293.15
0.014000       2.0641E-5      -4.1260E-6     -4.1279E-6     5.1599E5       -2.7564        -1.1205E-8     293.15
0.015000       2.0403E-5      -4.0777E-6     -4.0801E-6     5.1001E5       2.7821         -7.9614E-8     293.15
0.016000       2.0162E-5      -4.0295E-6     -4.0319E-6     5.0399E5       -2.7882        -2.1886E-8     293.15
0.017000       1.9922E-5      -3.9812E-6     -3.9839E-6     4.9799E5       -2.7859        2.7387E-8      293.15
0.018000       1.9684E-5      -3.9328E-6     -3.9361E-6     4.9201E5       2.7712         -1.0598E-7     293.15
```

Figure 13: Exported data from COMSOL

4.2 Using FBG-SimPlus

Any FEM export data file that fits the format outlined in section 3.1 of this user manual will work with FBG-SimPlus. One of our goals moving forward is to use an open-source FEM solver that can be manipulated by users in real time to obtain a FBG array simulation with less overhead.

In the first section of FBG-SimPlus labeled “Select Stressed/Strained Path Files”, select the “Add Files” button and navigate to the .txt file exported from COMSOL. Notice how 8 lines of header information exists in this text file before the extracted data is actually presented. Modify the “Skip Rows” parameter to reflect this by entering 8. The “Path Distance Input Units” should already be in the SI unit of meters, so also select this option.

In the second section of FBG-SimPlus labeled “Type of Simulation (Contribution)”, select Longitudinal Strain, Uniform and Non-Uniform, Transverse Stress [Pa. To give the most realistic simulation based on our data input, we recommend always selecting this combination of options. Of course, it is up to the user to decide if they would like to view certain contributions individually. Because the input data from the COMSOL model had body temperature of 292.15 K, for this particular tutorial, we would like to see the effect of raising the body temperature to 100 C, or 373.15 K. To do this, select Emulate Model Temperature [K] and enter 373.15 into the box. The Host Thermal Expansion Coefficient is the value for the Concrete material used in COMSOL. Navigate back to COMSOL under “Component 1 -> Materials -> Concrete” and observe this value to ben 10E-6 [1/K]. Back in FBG-SimPlus, enter 10E-6.

In the third section of FBG-SimPlus labeled “Simulation Parameters”, leave all of the following parameters except change the Simulation Resolution [nm] to 0.1. This will decrease to the total computations in our simulation by a factor of 2. The Fiber Attributes predefined in this section are for a Silica Glass fiber, one of the most widely used materials for FBG sensors.

In the fourth section of FBG-SimPlus labeled “Virtual Fiber Bragg Grating Array Configuration”, increase the Number of FBG Sensors to 3. This is how many virtual FBGs we intend to place along the predefined path. The FBG Length [mm] and Tolerance [mm] can be left at 10.0 and 0.01 respectively. For the FBG Position(s) section, click the add button and enter the following numbers in order: 20, 45, 70. This step sets the start position of where each virtual FBG will be placed. The relative location of each FBG is now defined as follows. FBG 1 from 20-30mm, FBG 2 from 45-55mm, and FBG 3 from 70-80mm along the path. For the FBG Array Original Wavelengths section, click on the Auto button. This will automatically divide our waveband and assign an initial Bragg wavelength of 1525nm, 1550nm, and 1575nm to FBG 1, FBG 2, and FBG 3, respectively. Usually, FBG manufacturers characterize FBGs by their initial grating period. We have characterized them by their directly related Bragg wavelength here for convenience.

Now in the fifth section of FBG-SimPlus labeled “Spectrum Simulation” check the box “Plot the undeformed FBG reflected signal” and click the Generate button. The computation may take a minute or two but it will be apparent once the computation has completed by the status bar reaching 100%. The FBG-SimPlus interface should look like figure X once all of these steps have been completed.

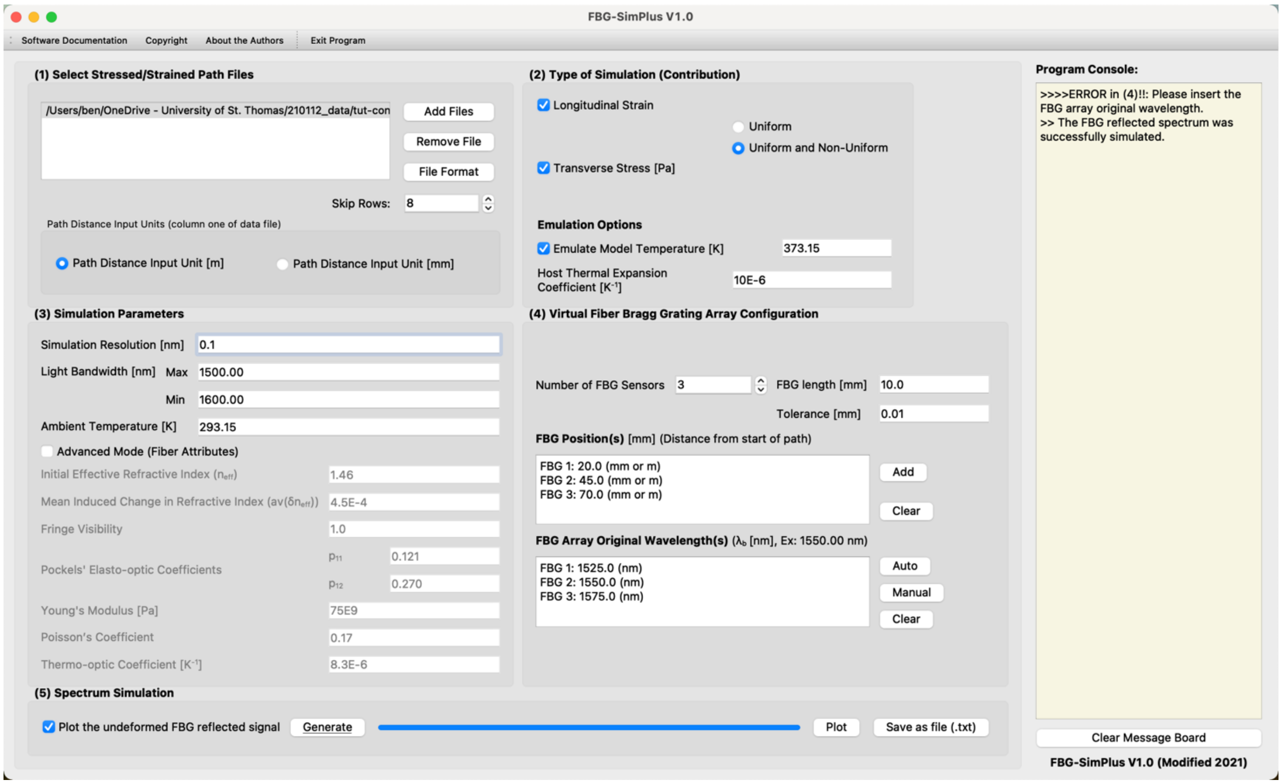


Figure 14: Running FBG-SimPlus to generate a reflected spectrum

4.4 Spectrum Simulation Results

To view the reflected spectrum of the FBG sensor array, click on the Plot button. The window in figure X will be displayed. Feel free to change of the graphical settings of the program, such as selecting the Legend checkbox to view more information about what each waveform represents.

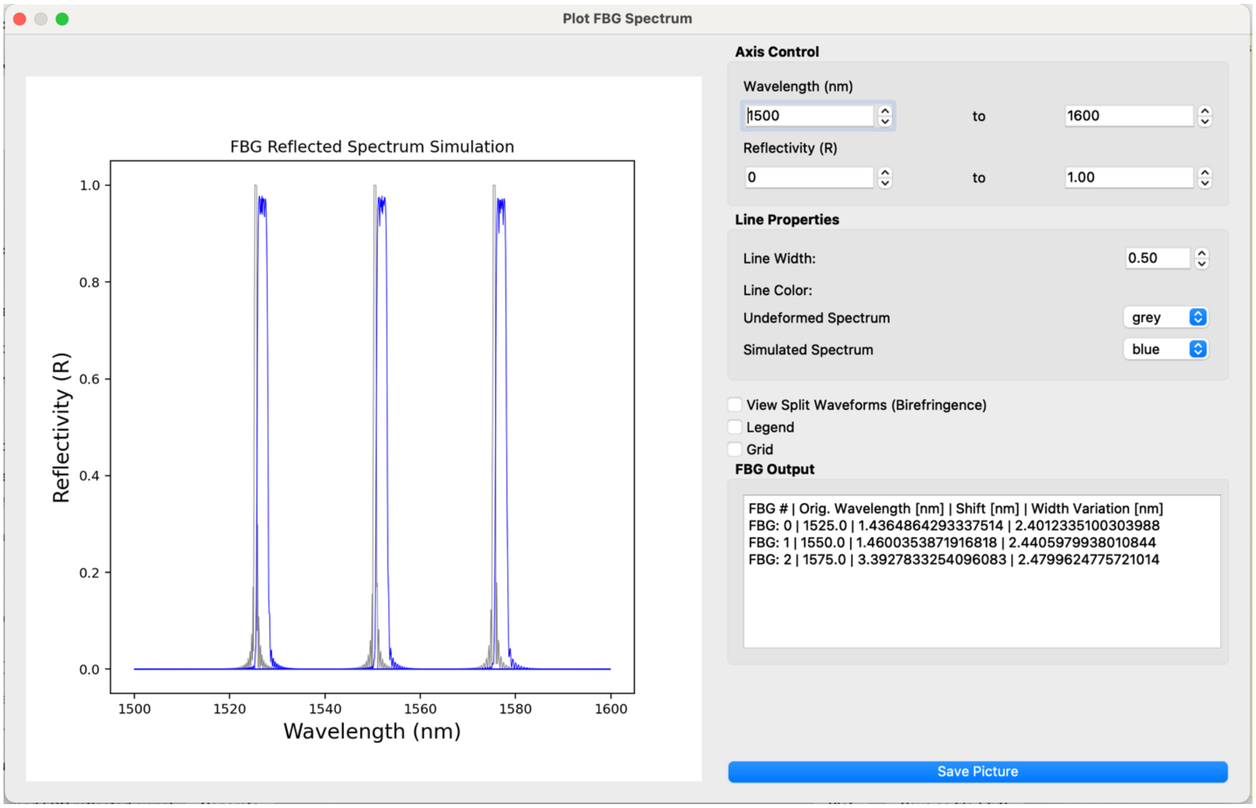


Figure 15: Plot of reflected spectrum for virtual FBG array

The exported COMSOL data and model have been included in this repository under the "tutorial" directory.