Here is a tutorial in importing SolidWorks into ANSYS and the steps needed to apply a Gaussian Force to the LIGO test mass that’s imported.

Using SolidWorks:

1. Download the SolidWorks Zip Folders from the DCC
	1. <https://dcc.ligo.org/login/index.shtml?entityID=https%3A%2F%2Fdcc.ligo.org%2Fshibboleth-sp&return=https%3A%2F%2Fdcc.ligo.org%2FShibboleth.sso%2FLogin%3FSAMLDS%3D1%26target%3Dss%253Amem%253A15b6c314d87e3fa8b3768d89cb6b9836fe39c754> (LIGO-D1000760-v4)
	2. Open/export the zip folders into a different file
2. For only the test mass and ears, open the file called: D0902456 ITM OPTIC WITH EARS ASSEMBLY. This will represent the bottom mass.
3. An assembly can be imported into ANSYS but it’s easier to convert an assembly into a part before importing it into ANSYS. Also this will allow you to perform a split line.
	1. To do this, resave the assembly but before pressing save, change the file to PRT extension instead of Assembly.
4. Open the PRT file of the test mass, make a new sketch on the face of the test mass.
5. Draw a circle (Or whatever shape) on the test mass. In this case a 0.1 m circle was made in the middle of the test mass.
6. From there go to “Insert” and then press “Curve” and then go to “Split Line”. Split line allows a user to project the sketch onto the part. This makes it possible to press the sketch separately from the part. This is a useful tool because in ANSYS, once split line has been used on a sketch, a user can press that shape separately on the face of the object. The importance of this tool allows a user to click on the circle separately and apply a force in that particular area.
	1. In “Split Line” press the “sketch” you want to project onto the “face” of the object.

Importing SolidWorks into ANSYS:

1. Once the part has been made, choose the “static structural analysis”.
2. In the geometry: Import the test mass that was converted into a part
3. Once imported, press the geometry again and “edit”
4. In the edit module, “generate” the part and once the part has loaded notice the circle can be pressed such that it is a different part of the test mass surface.
5. Exit geometry module

Applying Gaussian Force:

1. Enter the “model” module to edit the setup of the analysis
2. Assign the material that you want for the test mass under “Geometry” 🡪 “Solid” under the “Material” and “Assignment”
3. Under the “Coordinate Systems”, make a new coordinate system
4. “Coordinate System:
	1. Click on the face, in this case the circle made on the test mass for the “geometry” and “apply”
	2. Under “Definition” 🡪 “Type” 🡪 Change from Cartesian to Cylindrical. This allows the Gaussian Force to be distributed correctly
5. Under “Analysis Settings” Turn on “Large Deflection”
6. Add “gravity” under “Static Structural”
	1. Treat the test mass is if it were the bottom mass so make sure gravity is pointing the correct way, such that the wires on the ears would extend upward.
7. Insert “Fix” to the top surfaces of the ears
8. Insert “Pressure”
	1. Press on the circle made to apply for the pressure
	2. Under “Definition” change the “Magnitude” to “Function” (the arrow at the end of the Magnitude entry bar)
	3. Once function has been activated: A Gaussian force of: 1/((3.141592654\*0.0156^2)\*2.718281828^((x/0.0156)^2)) was used in SI units. Note that ANSYS does not use symbols. Once that’s entered into the “Magnitude”, under “Function” the “Coordinate System” will appear. Change that to the coordinate system that was made in step 4 of the cylindrical system.
	4. Under “Graph Controls” Make sure that the X-Axis is changed from Time to X.
	5. Pick a range for the graph and the number of segments that you want to look at.
9. “Solve” the system and right click on “Solution” and evaluate the different results you need.