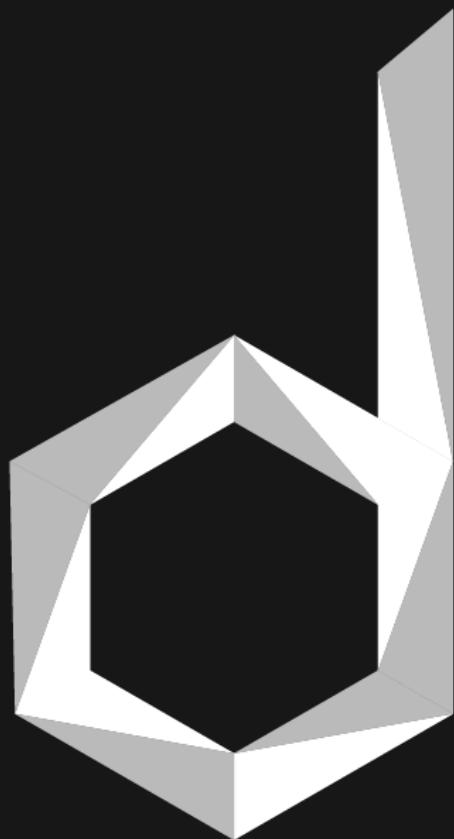


INGENIERÍA MECATRÓNICA



DI\_CERO

DIEGO CERVANTES RODRÍGUEZ

INGENIERÍA ASISTIDA POR COMPUTADORA

COMSOL MULTIPHYSICS 5.6

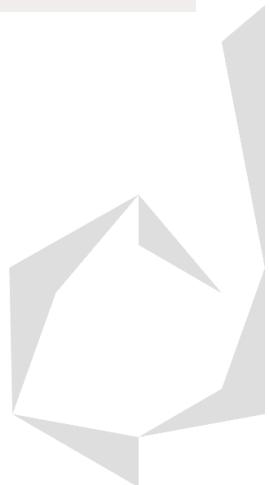
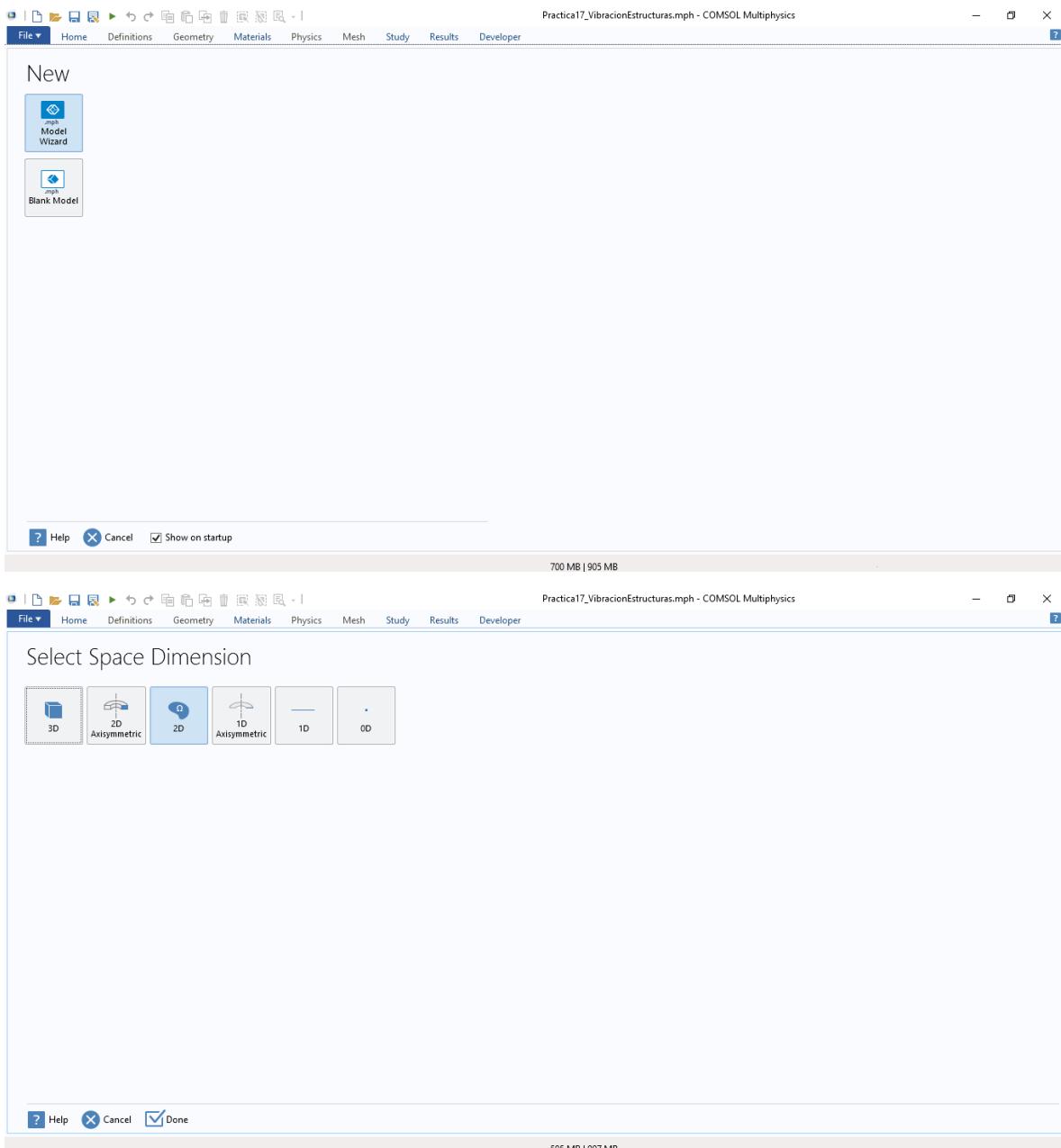
17: Vibración en Estructuras

## Contenido

CREACIÓN DE LA PIEZA EN COMSOL:.....	2
ANÁLISIS MECÁNICO EN COMSOL: .....	10
RESULTADO DEL ELEMENTO FINITO EN COMSOL: .....	21
ANÁLISIS DE VIBRACIONES ARMÓNICAS EN COMSOL:.....	29
GRÁFICA DE FRECUENCIAS ARMÓNICAS: .....	43
BIBLIOGRAFÍA:.....	53



# CREACIÓN DE LA PIEZA EN COMSOL:



**Select Physics**

The Truss interface is used for modeling slender elements that can only sustain axial forces. It can be used for analyzing truss works where the edges are straight, or to model sagging cables like the deformation of a wire exposed to gravity. It is available in 3D and 2D. Geometric nonlinearity can be taken into account. The material is assumed to be linearly elastic.

**Truss**

**Added physics interfaces:**

**Space Dimension**      **Study**

**? Help** **X Cancel** **Done**

**Select Study**

The Stationary study is used when field variables do not change over time.

**Stationary**

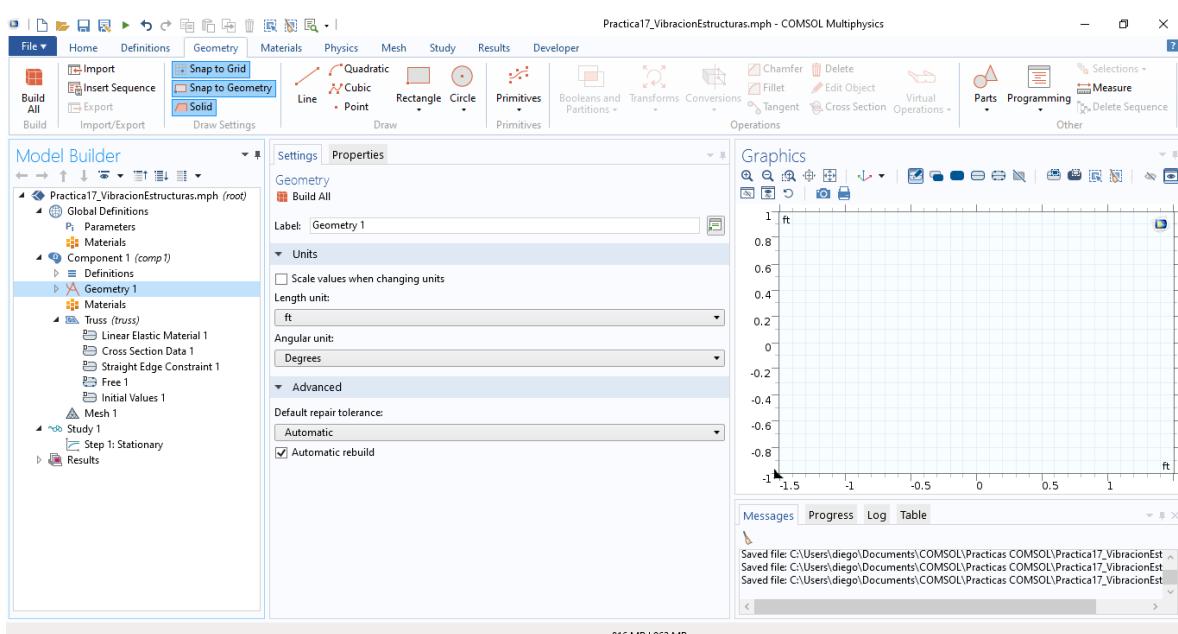
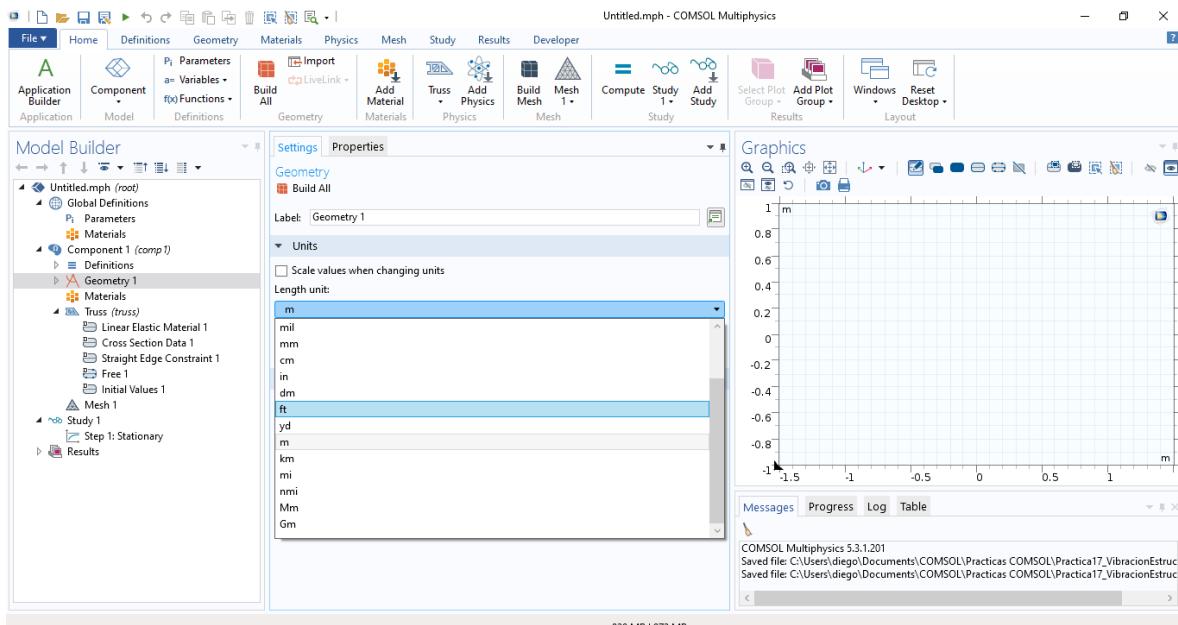
**Added study:**

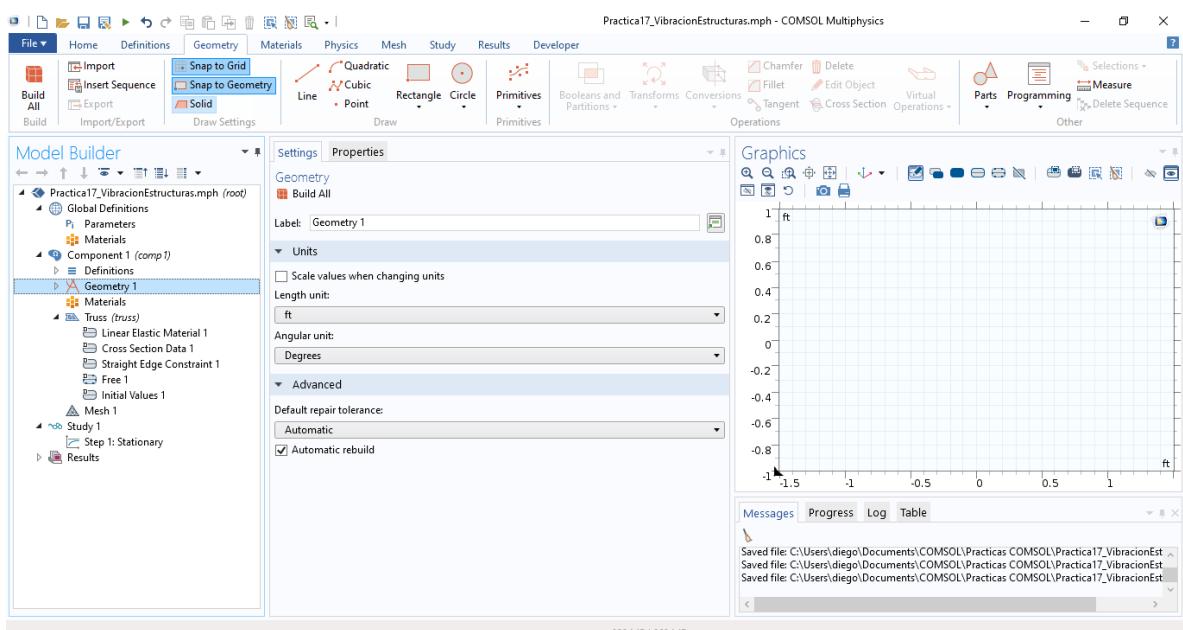
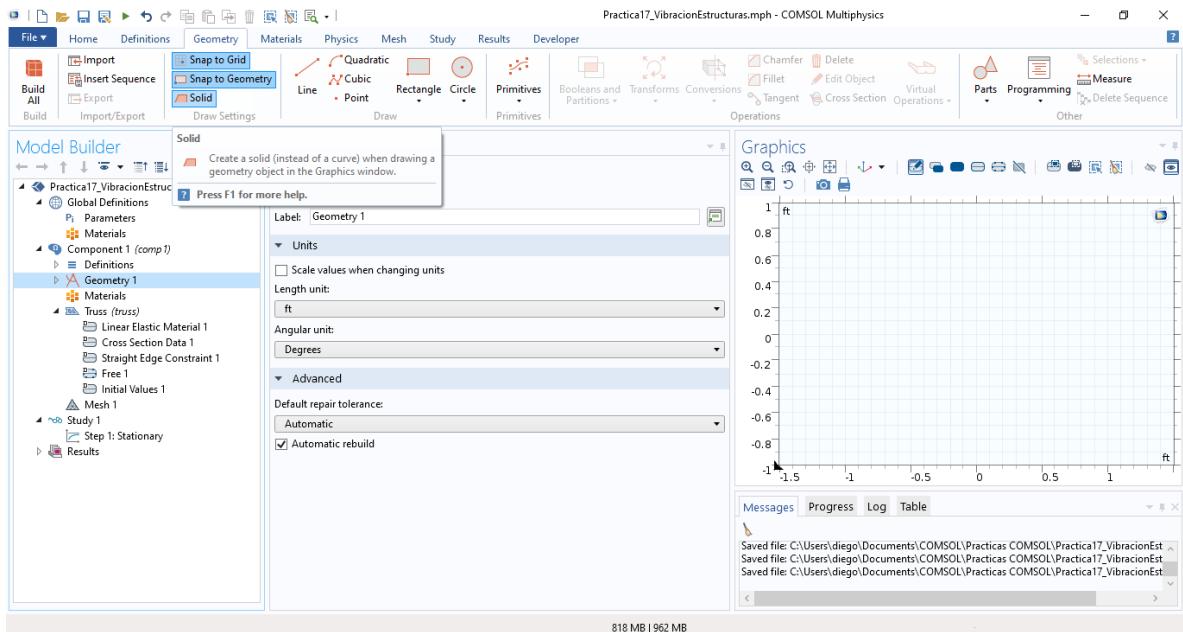
**Added physics interfaces:**

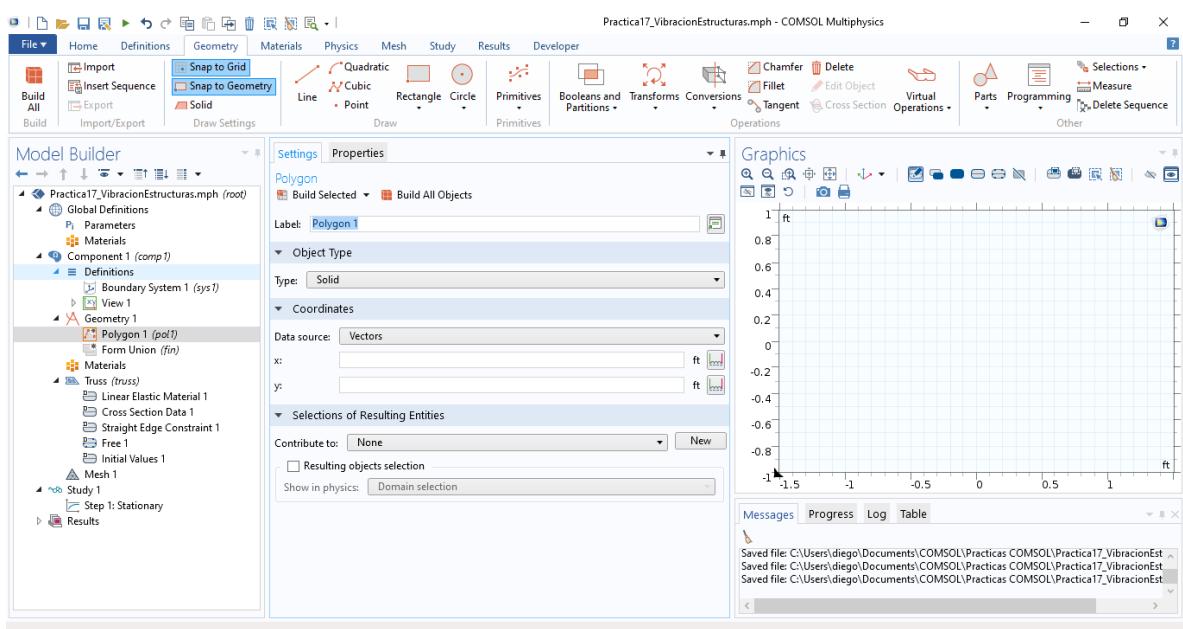
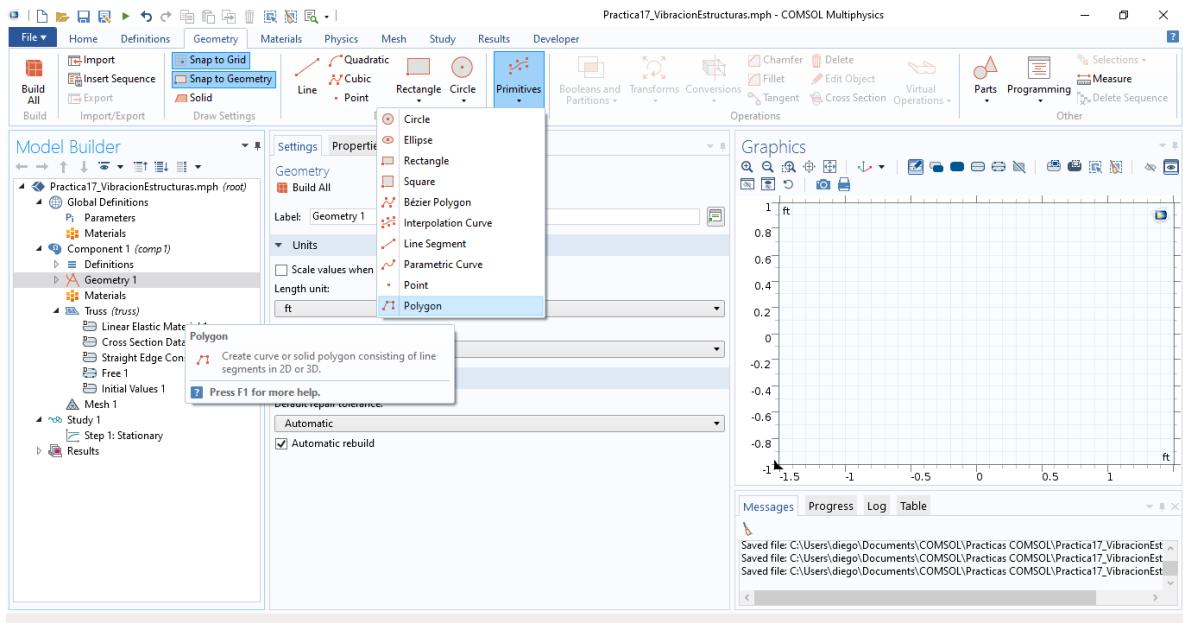
**Physics**

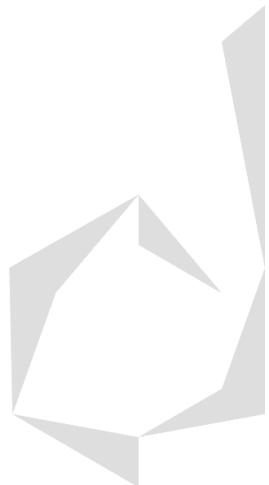
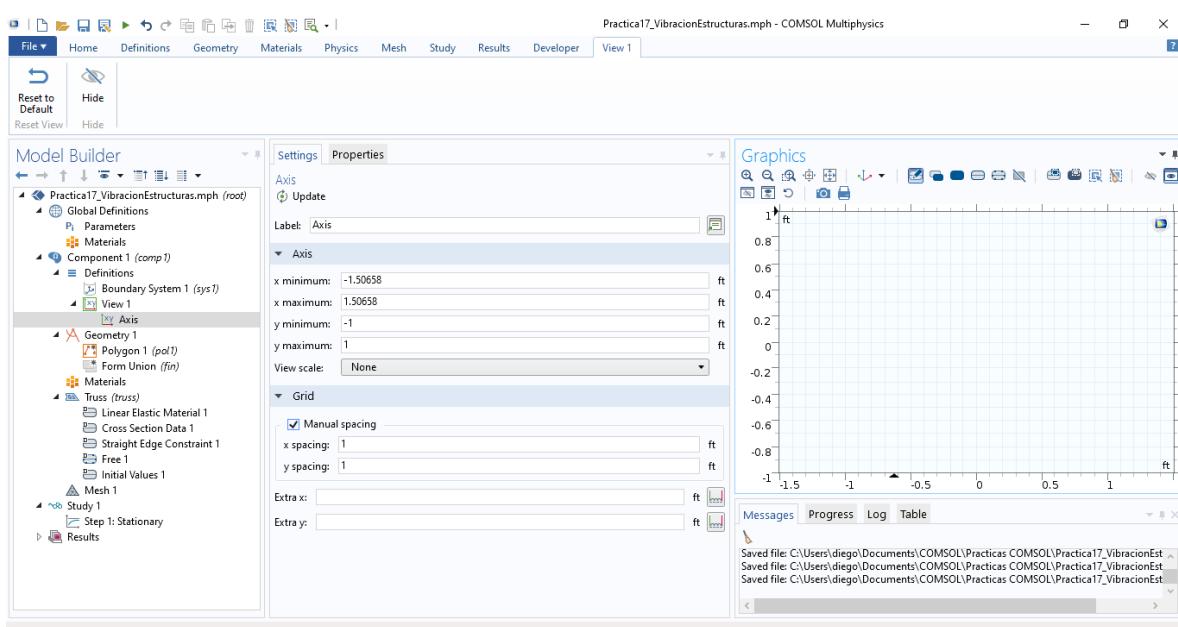
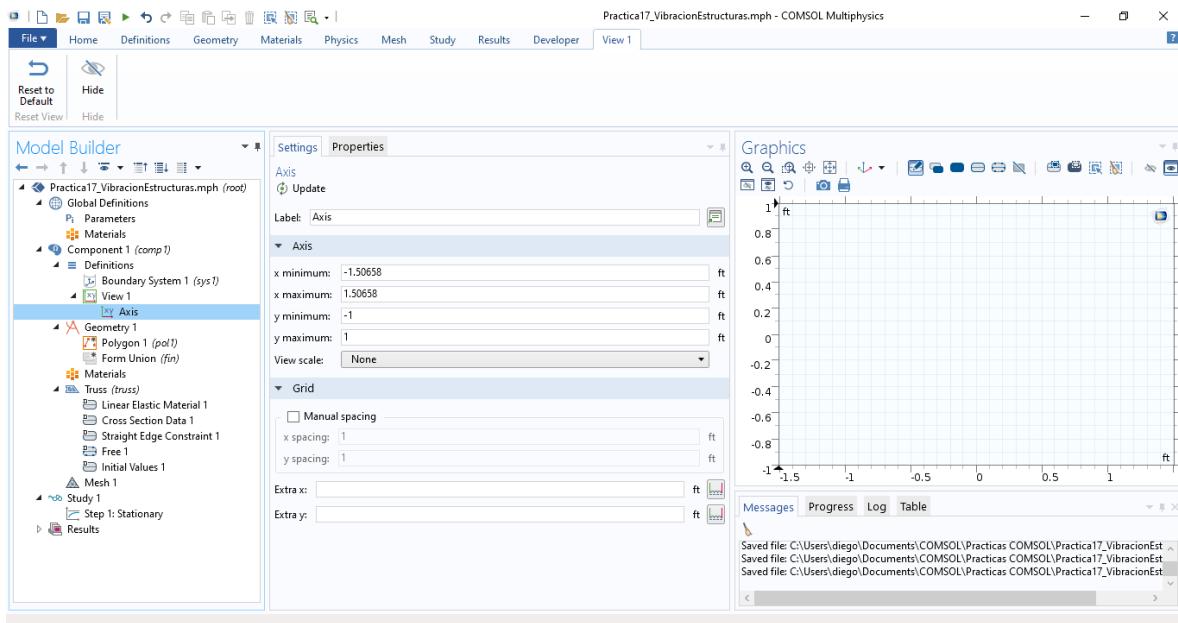
**? Help** **X Cancel** **Done**

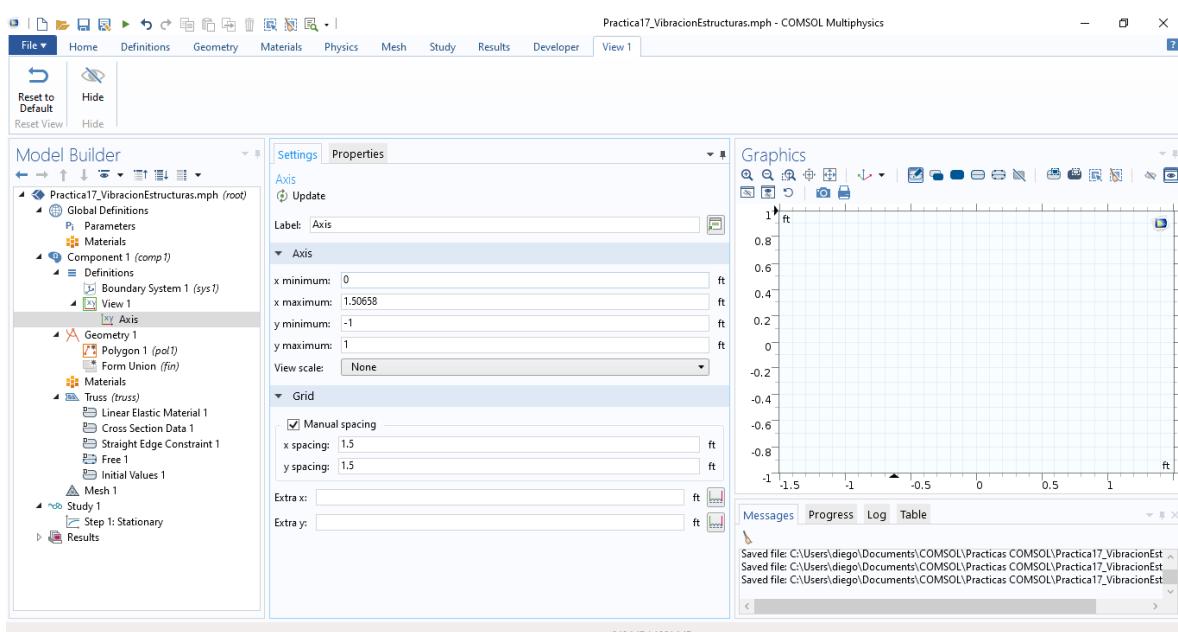
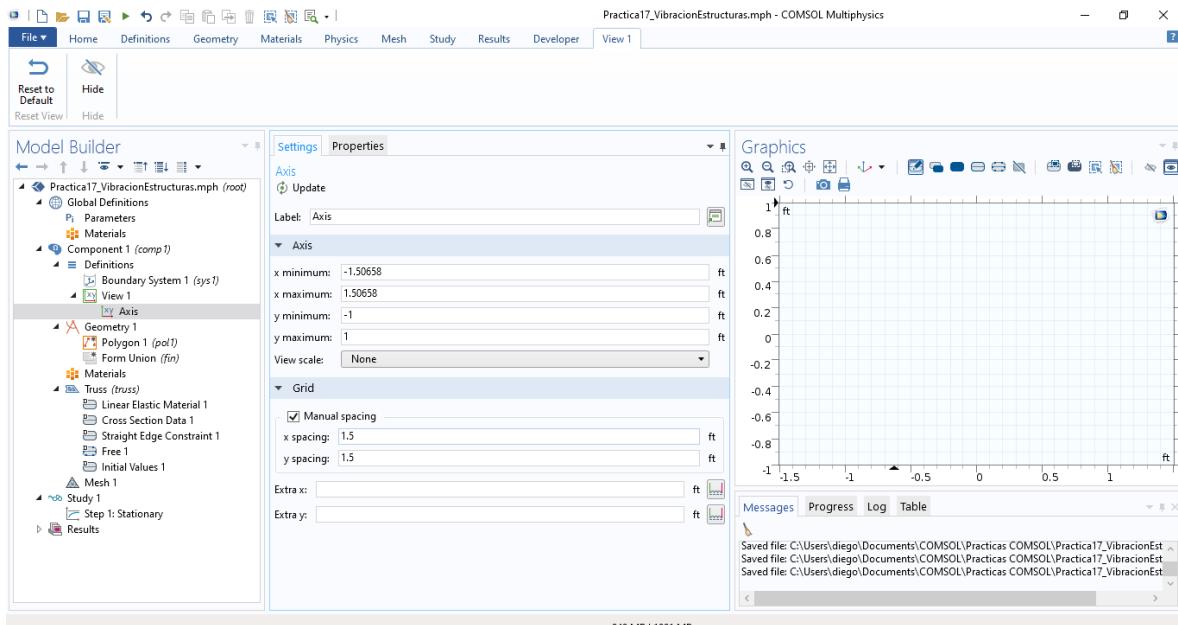


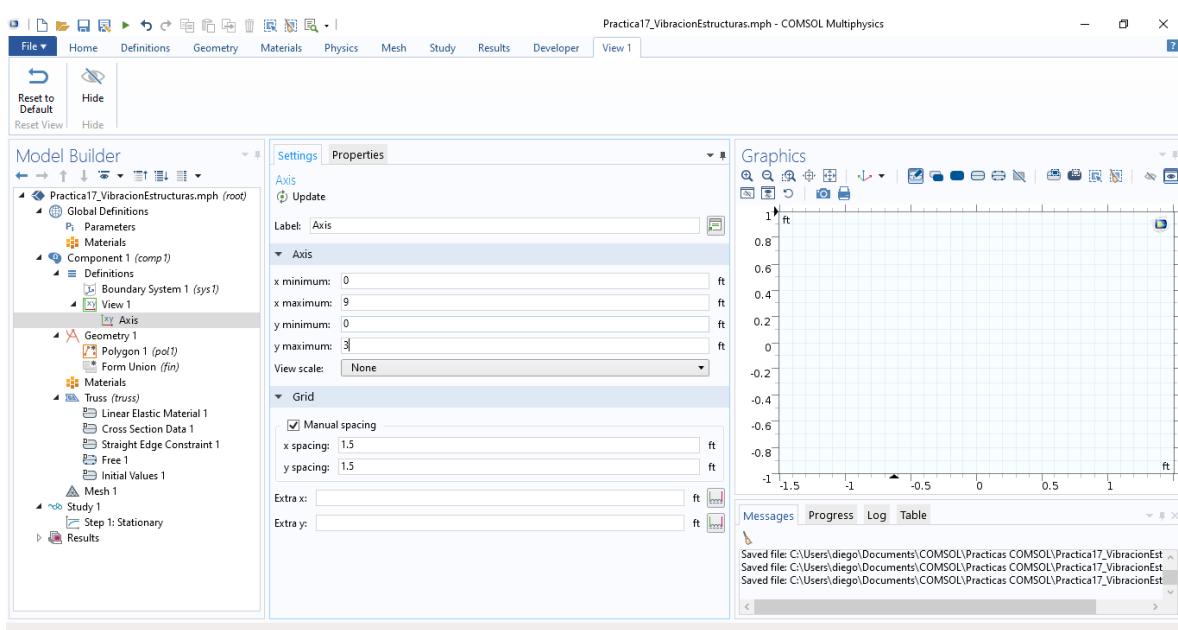
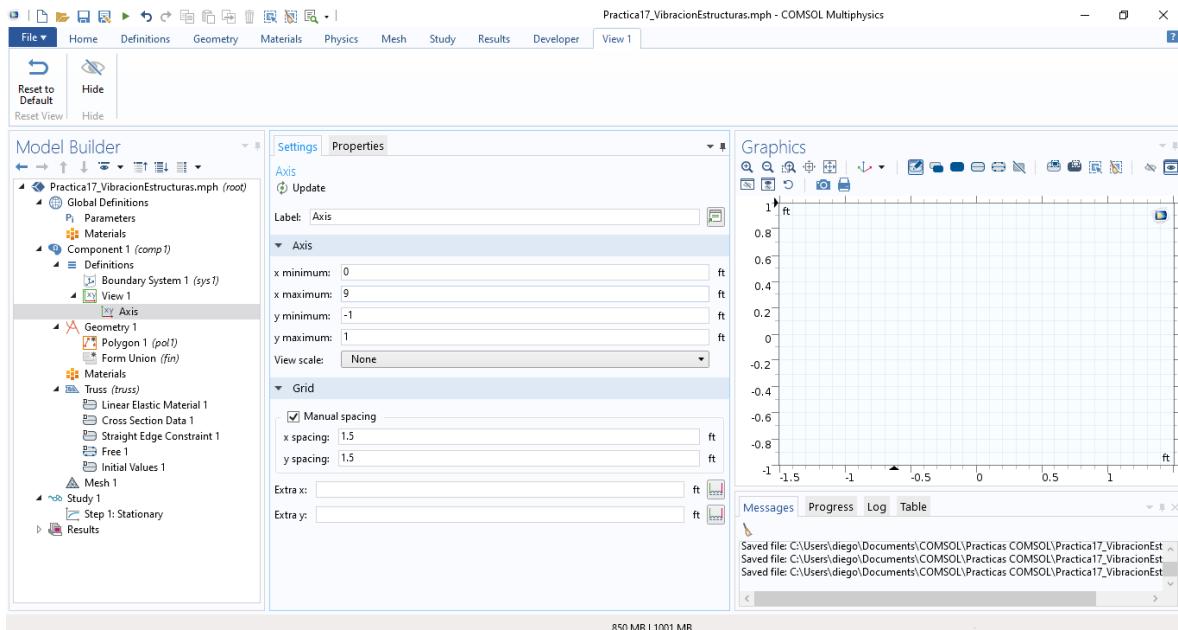




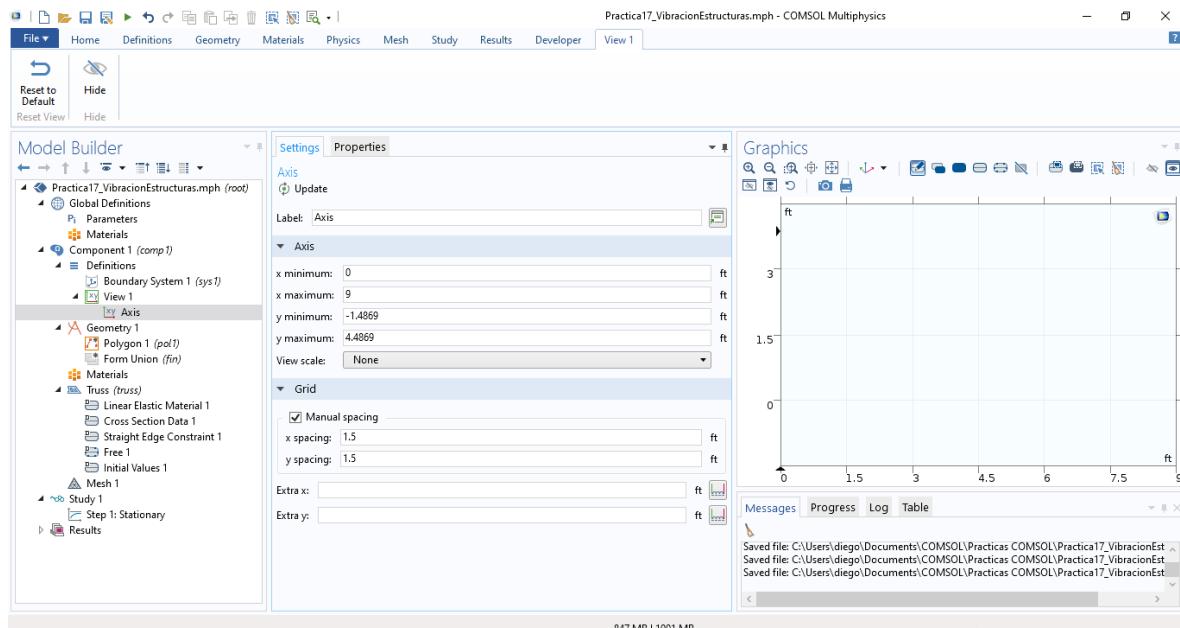




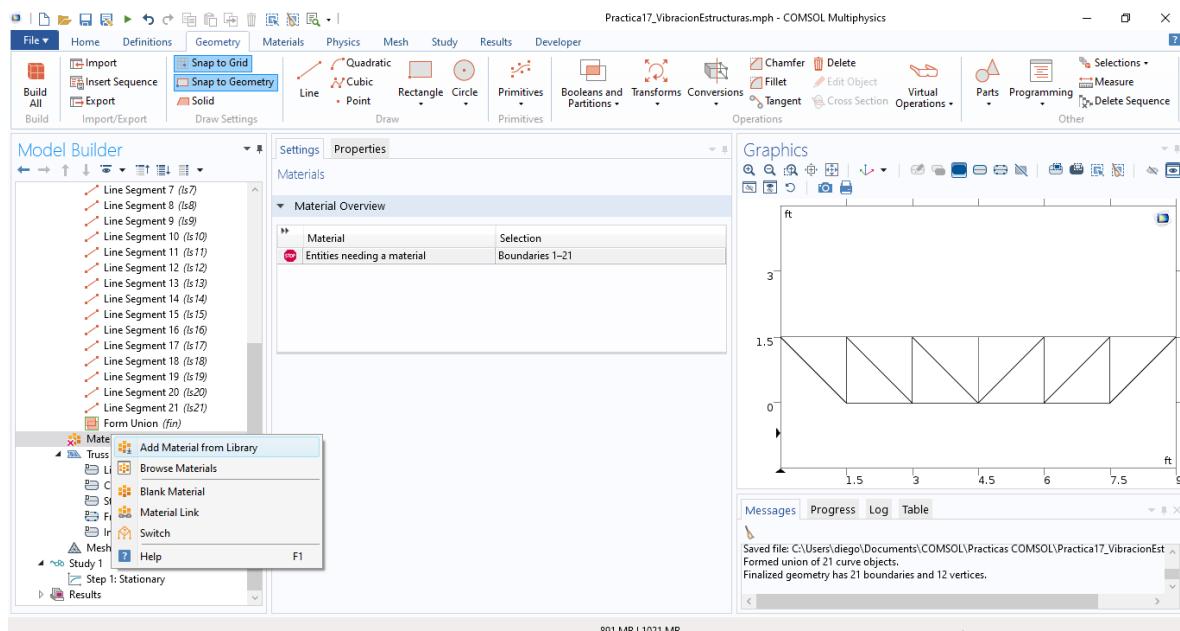


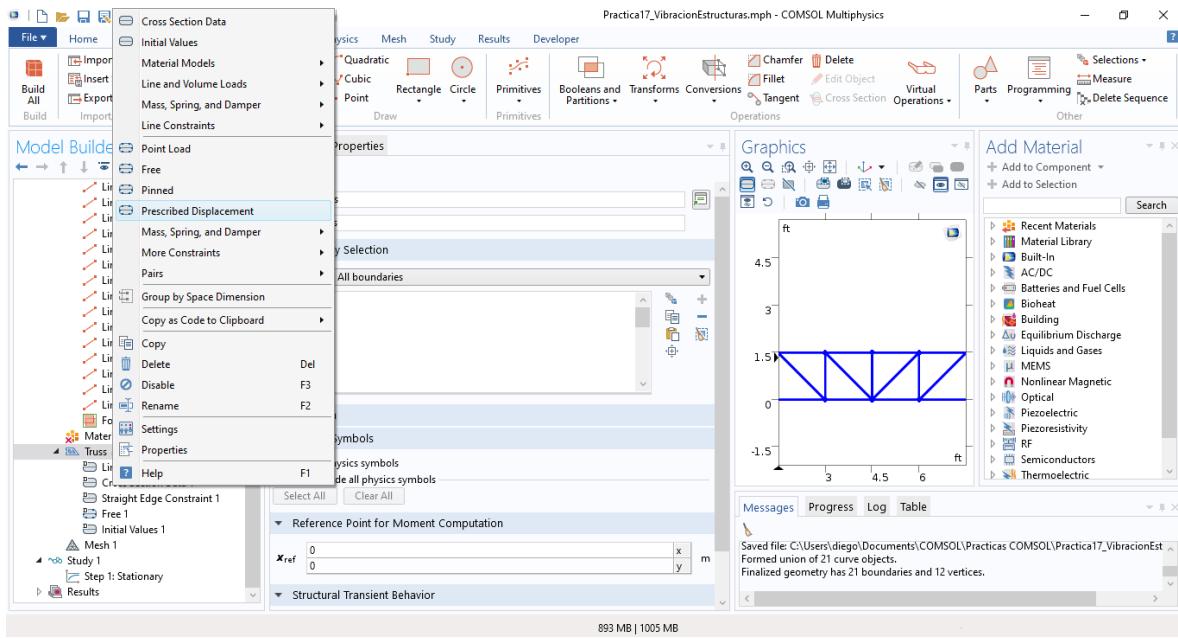


Esto se hizo para cambiar la forma en la que se muestra el display al mostrar la estructura.

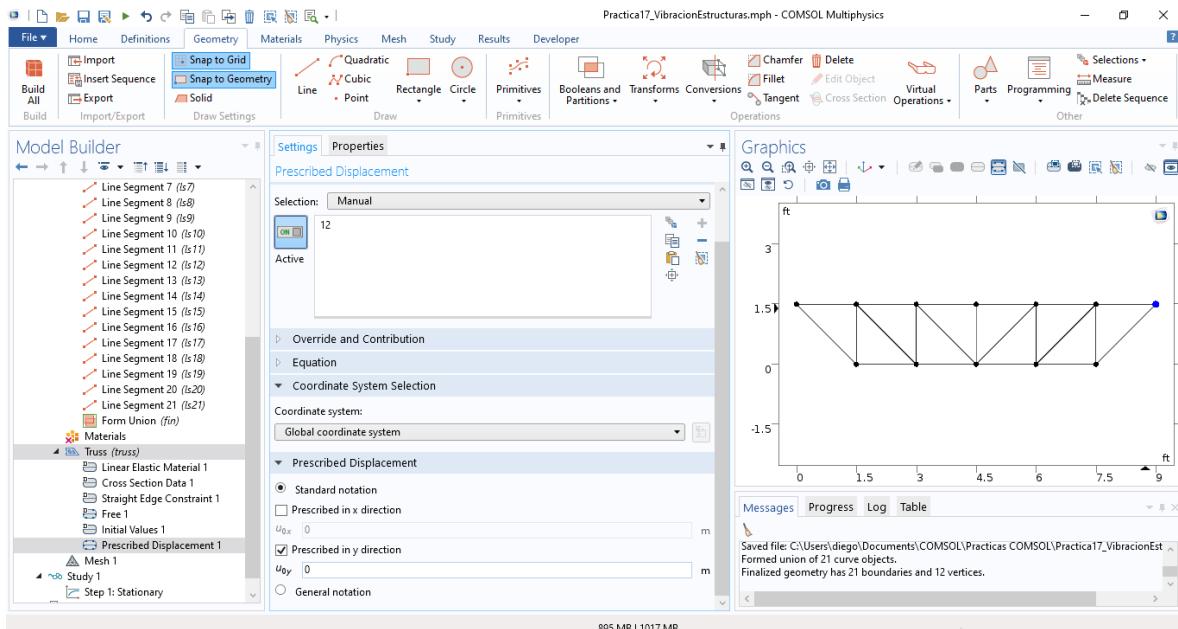


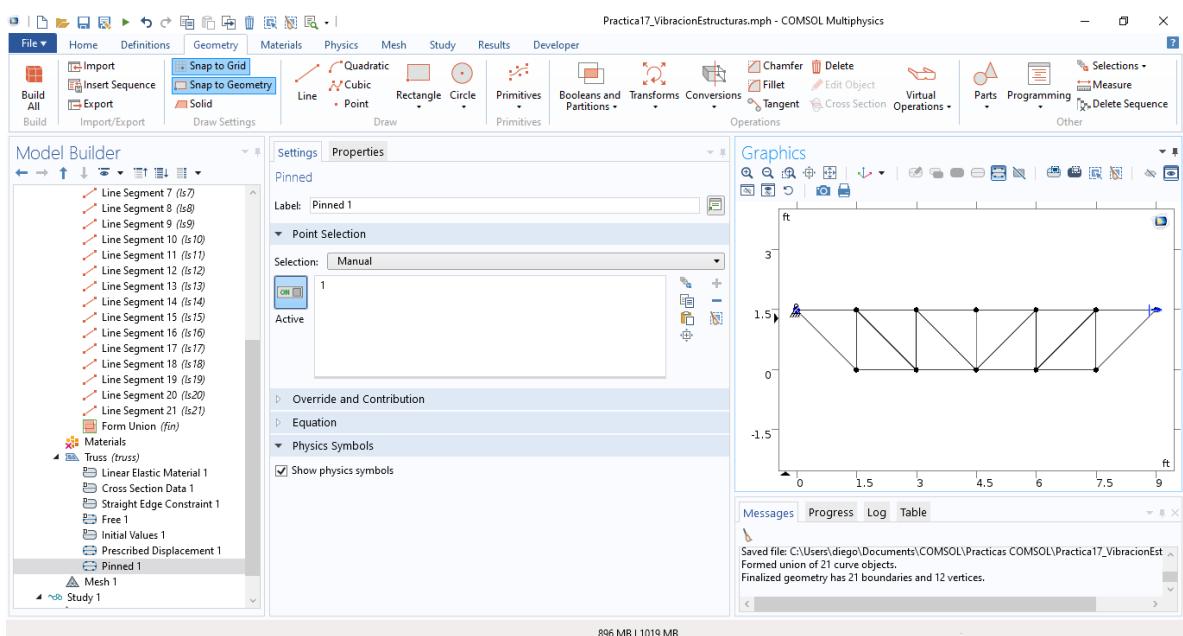
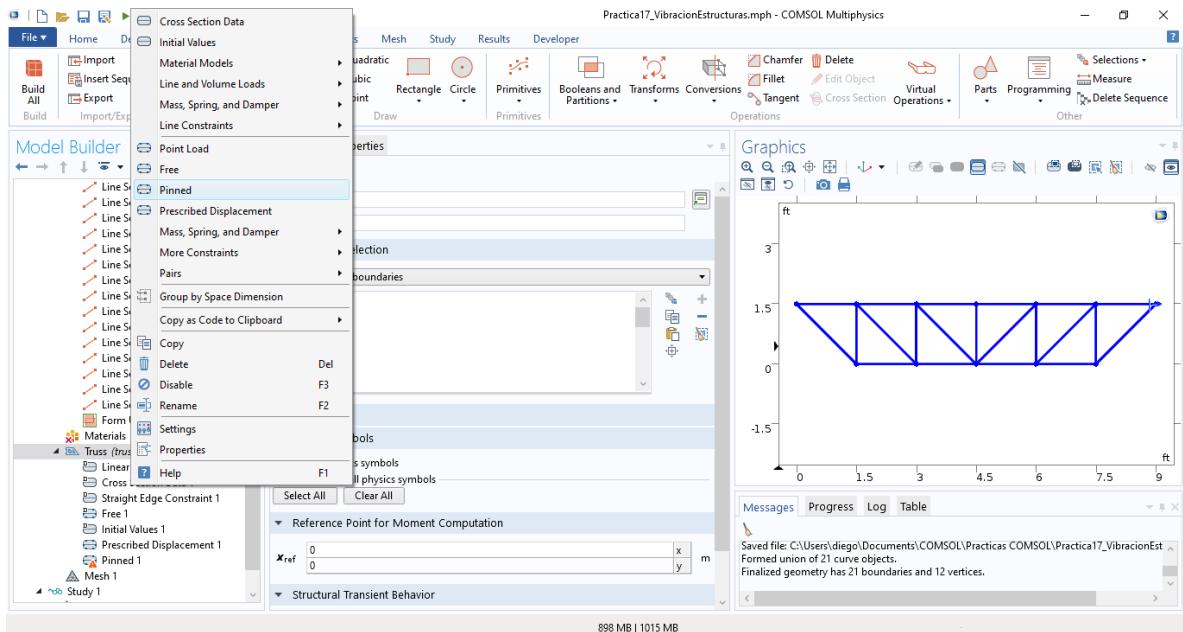
## ANÁLISIS MECÁNICO EN COMSOL:

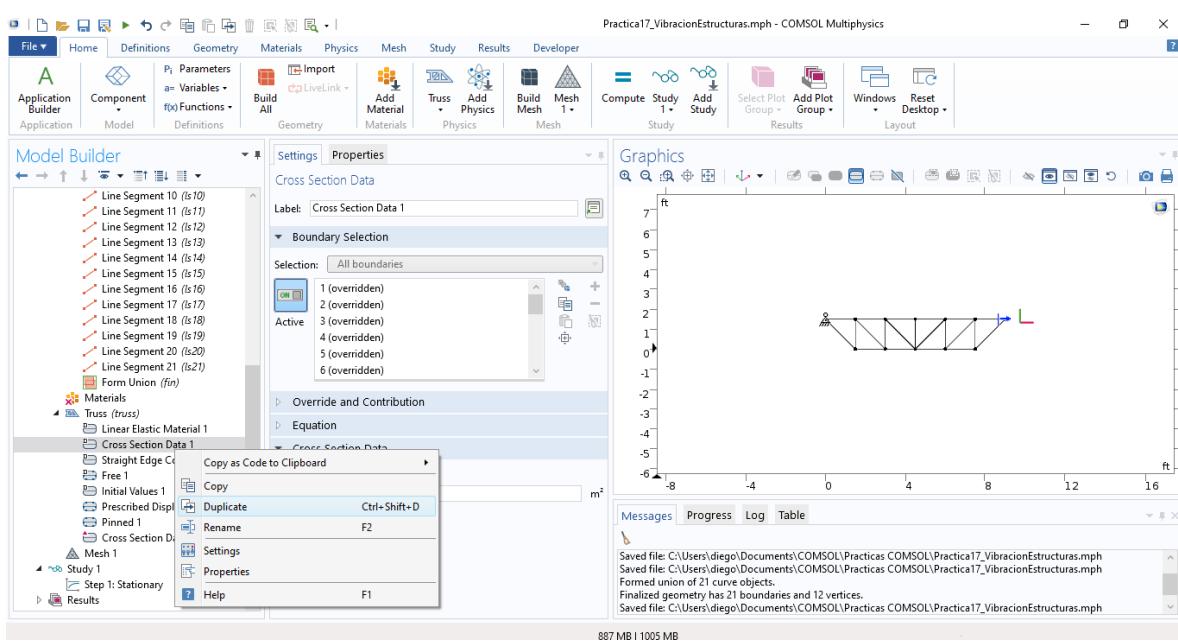
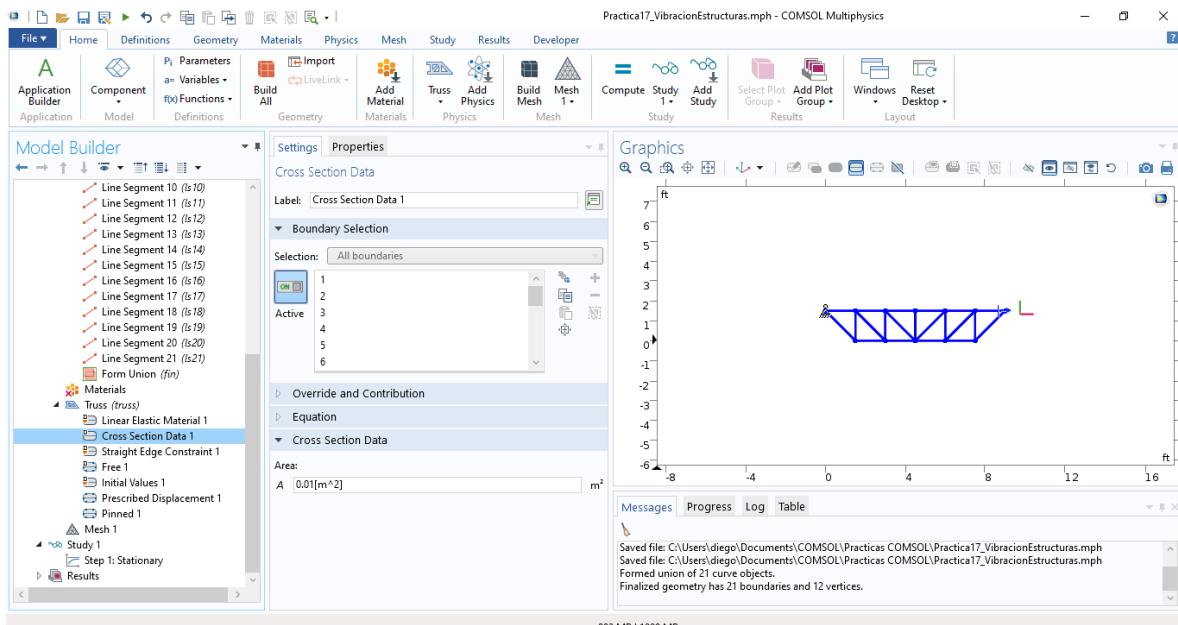


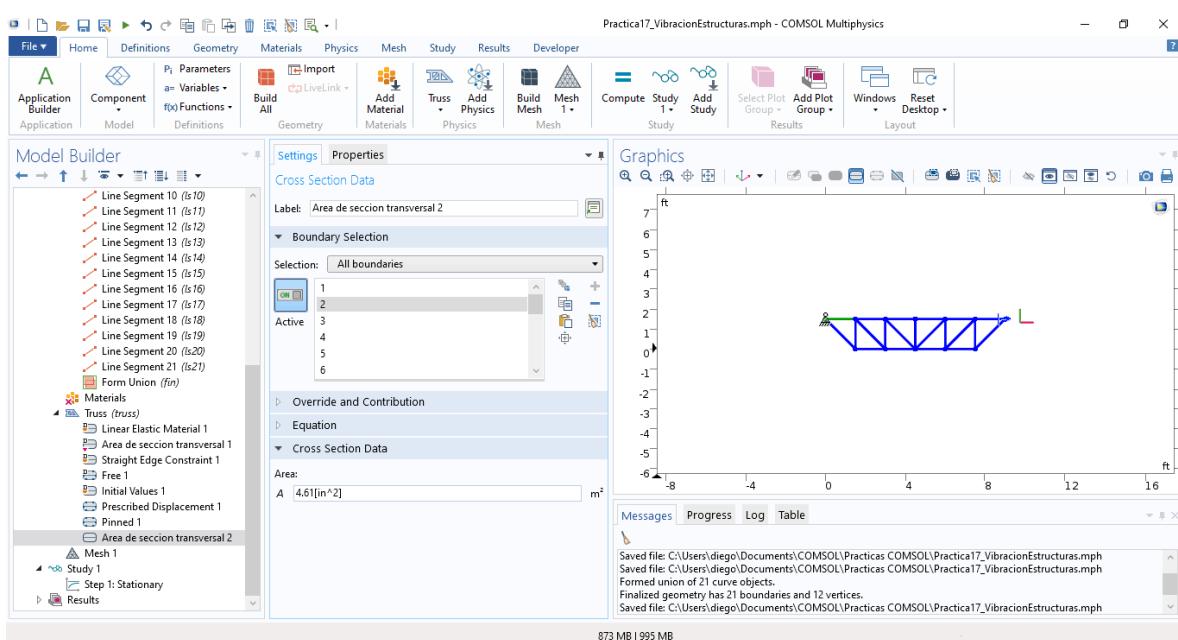
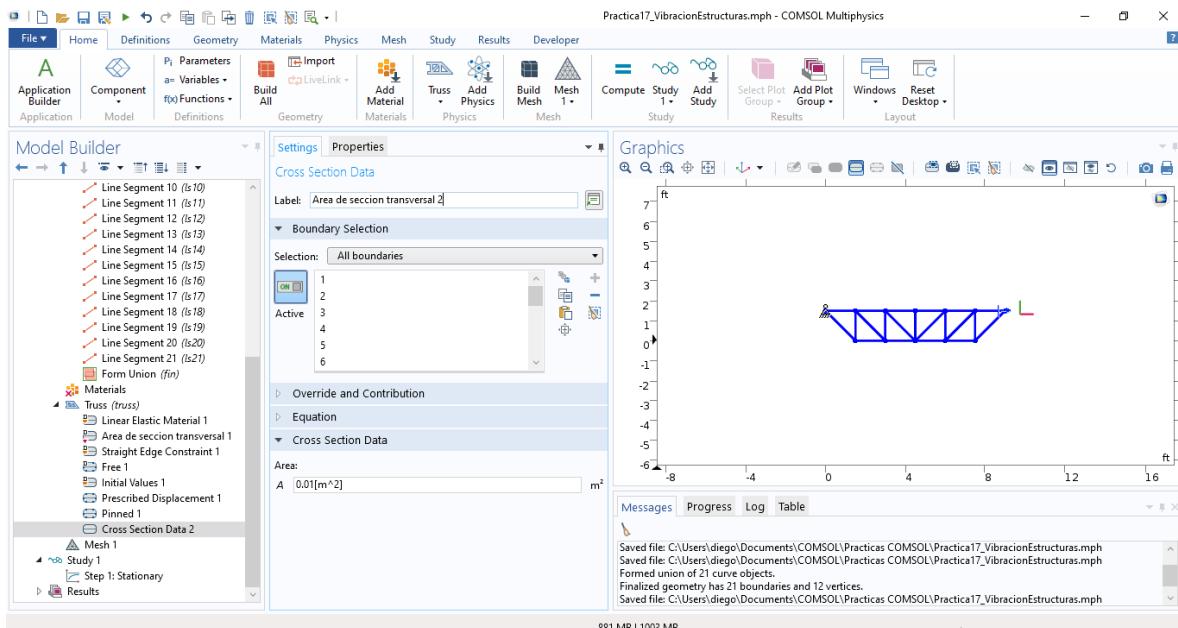


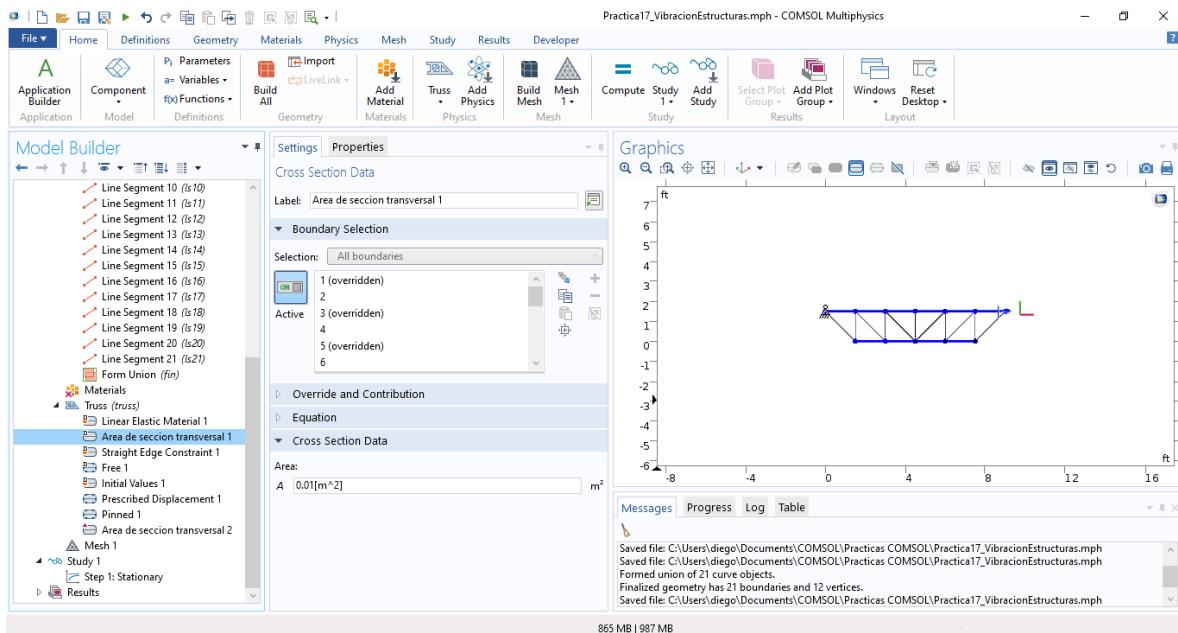
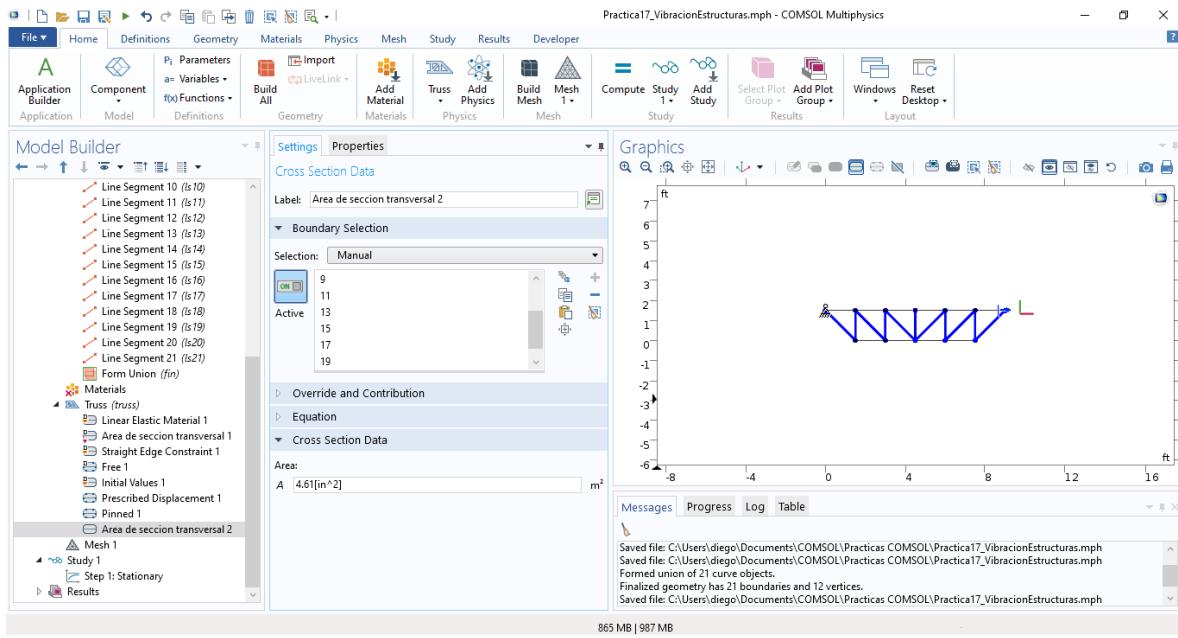
- **Prescribed Displacement:** Este apoyo no deja que se mueva verticalmente la armadura.



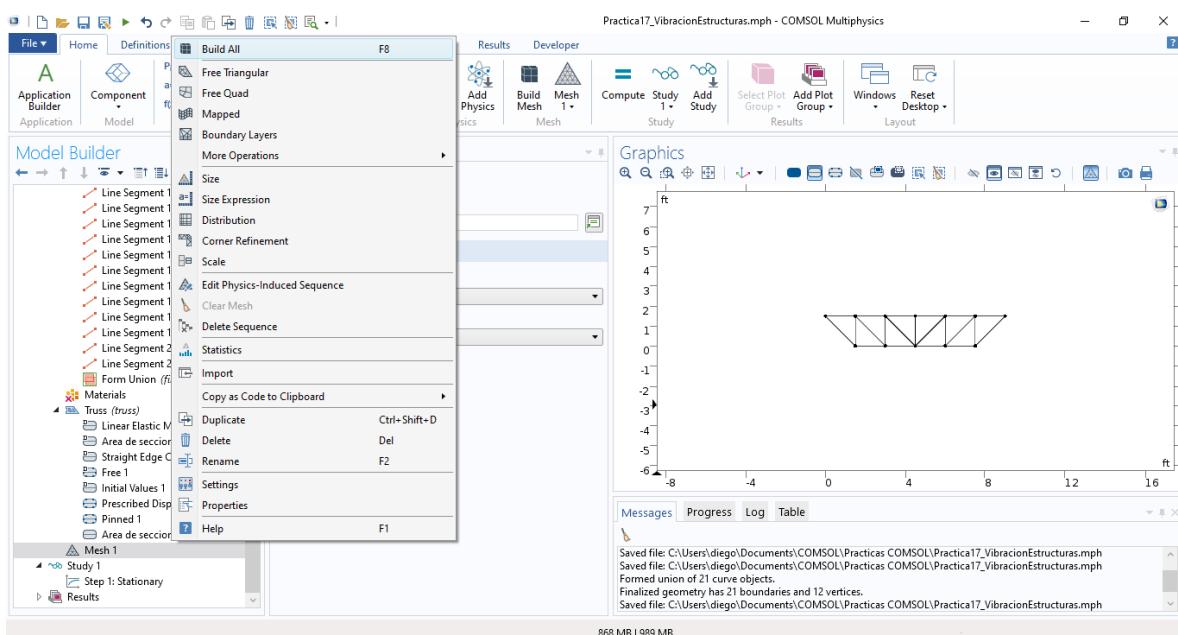
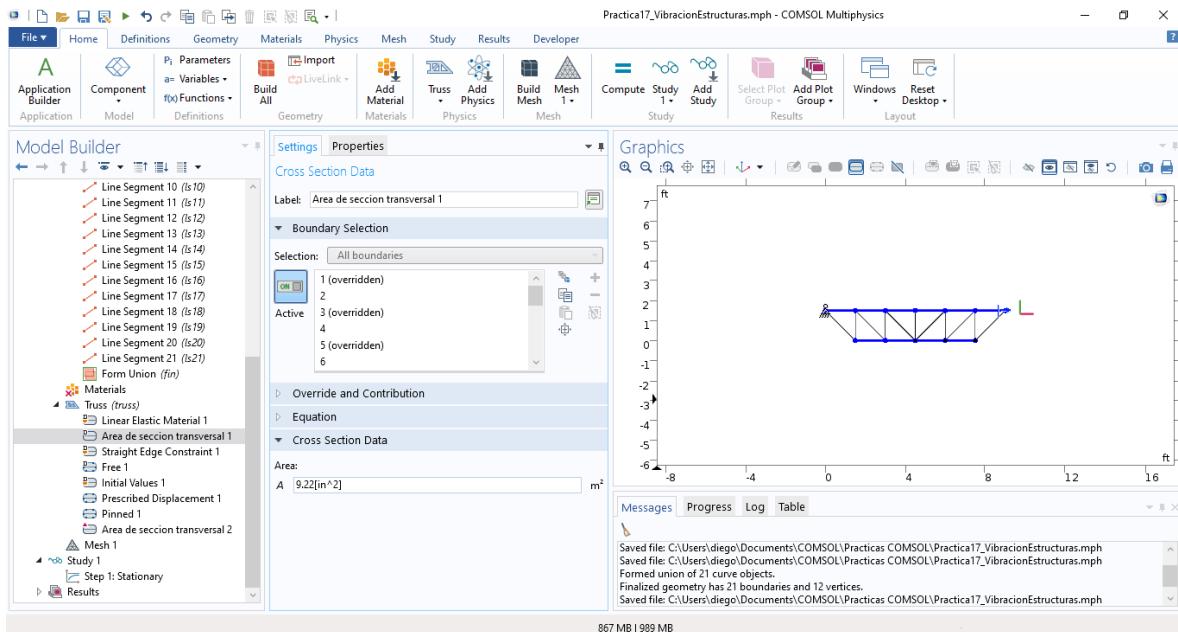


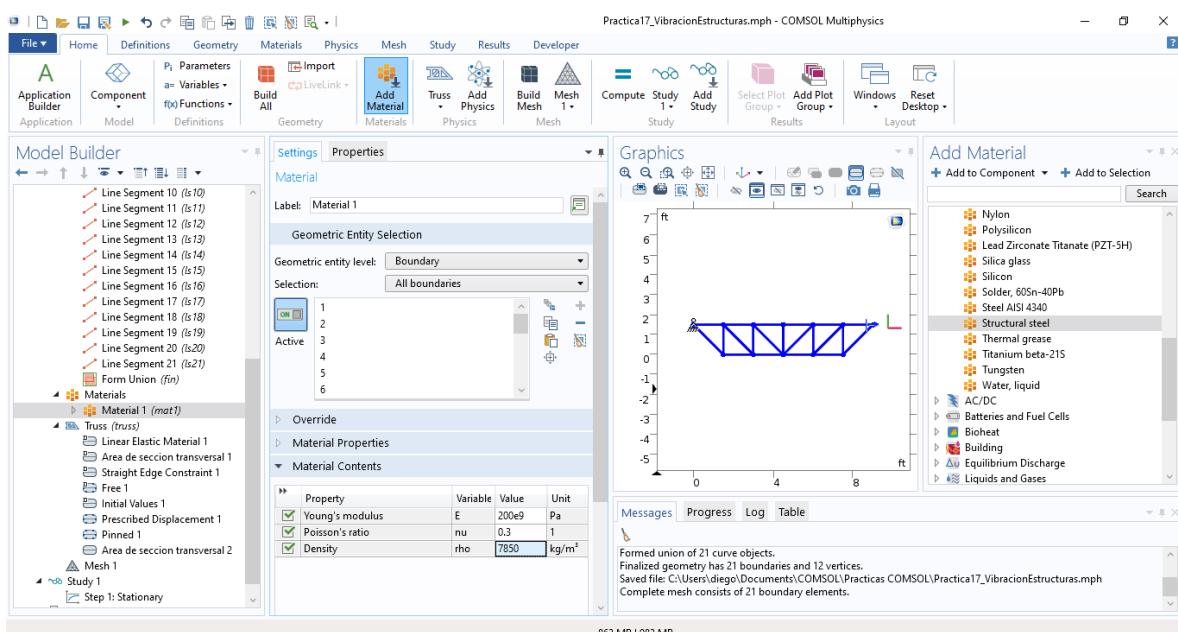
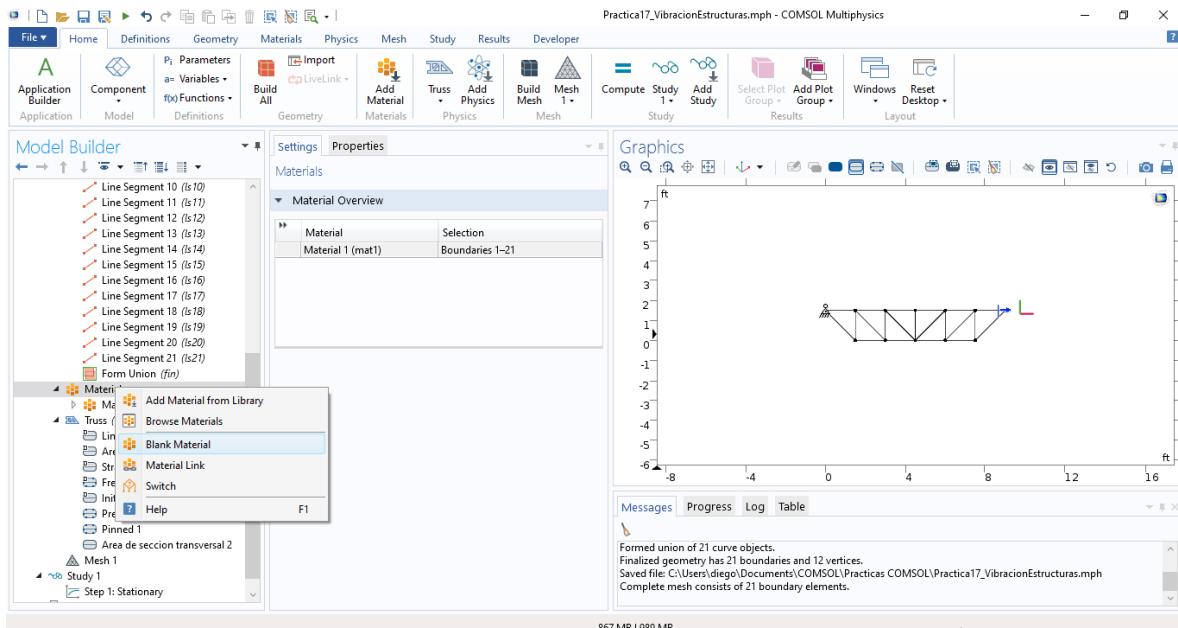


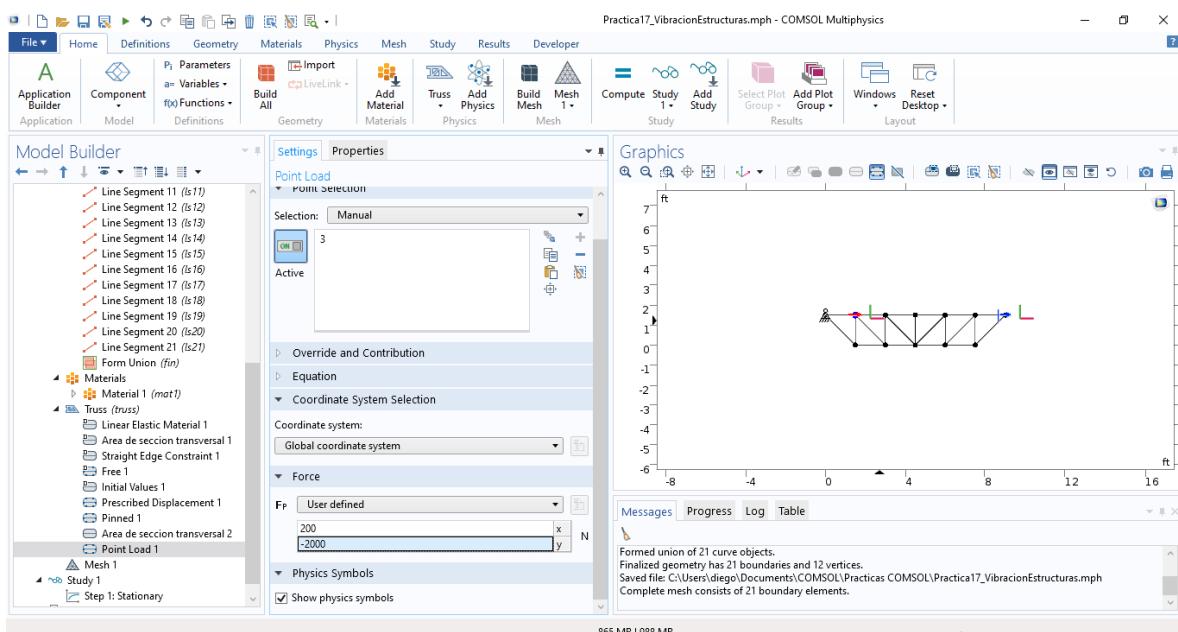
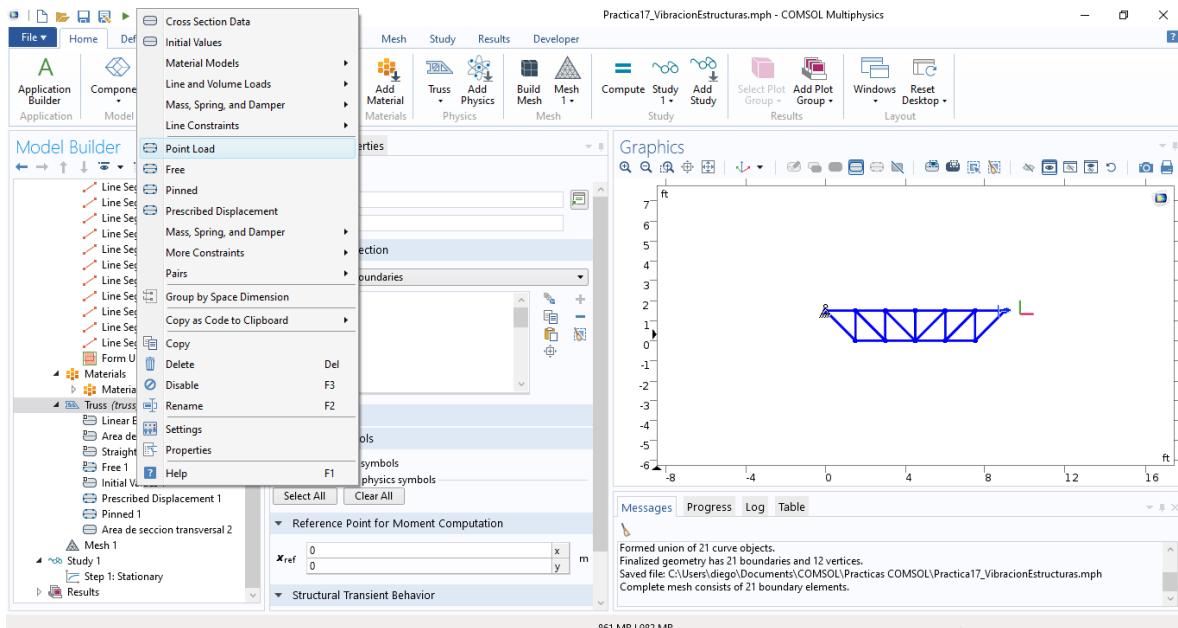


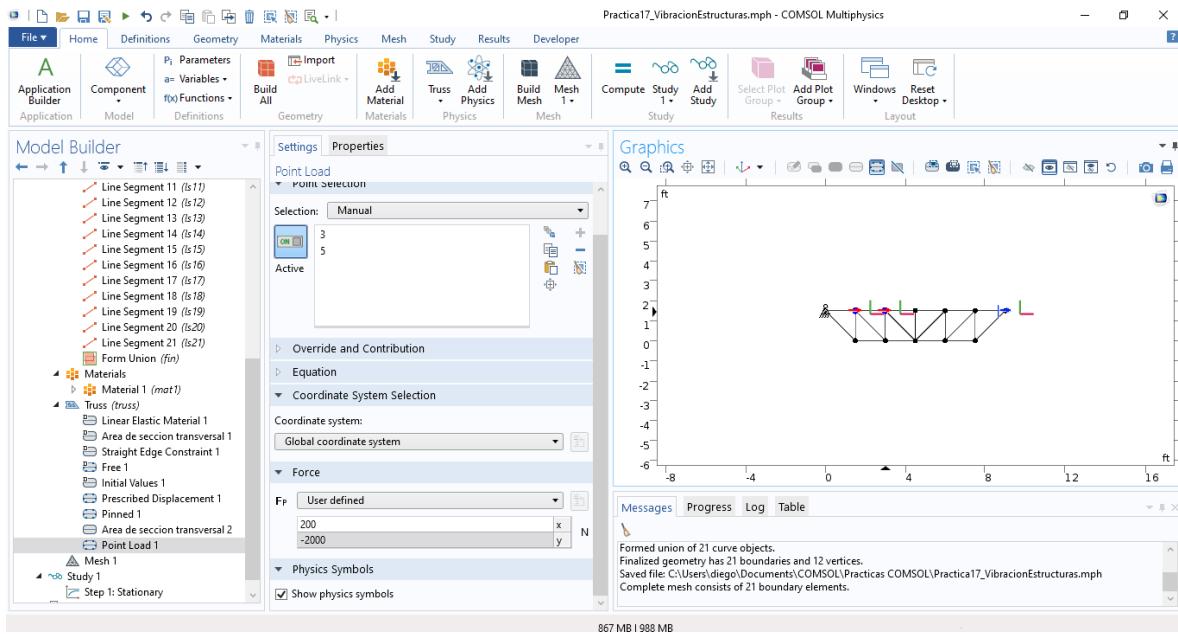


Va a haber 2 áreas de sección transversal distintas en la estructura, por eso se deben definir de la siguiente manera:

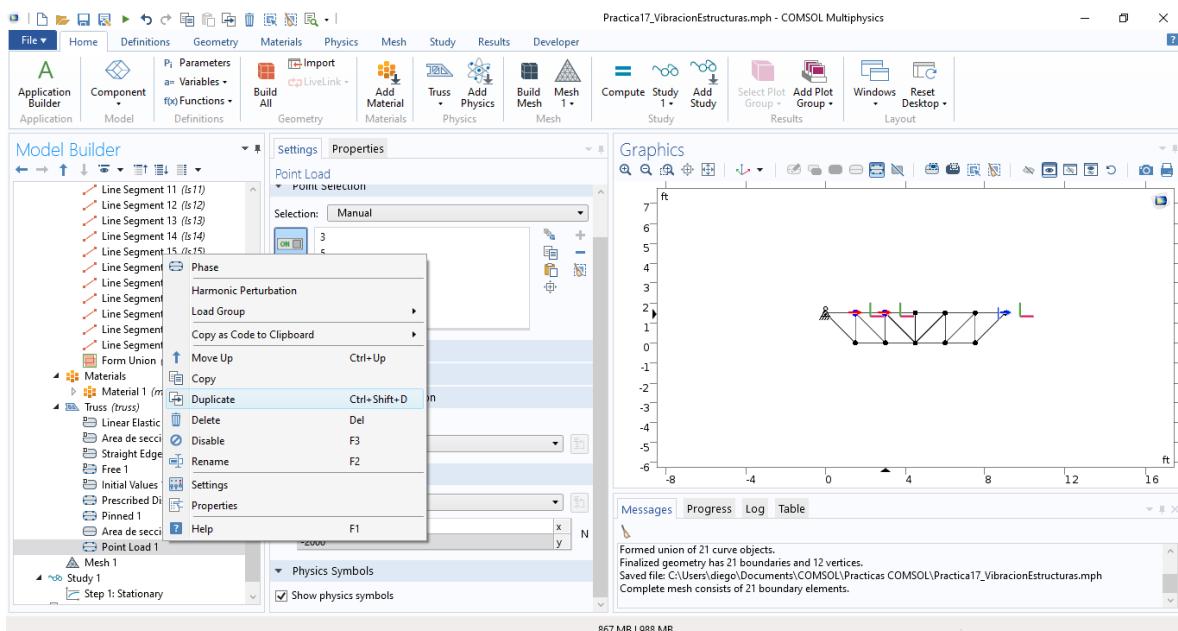


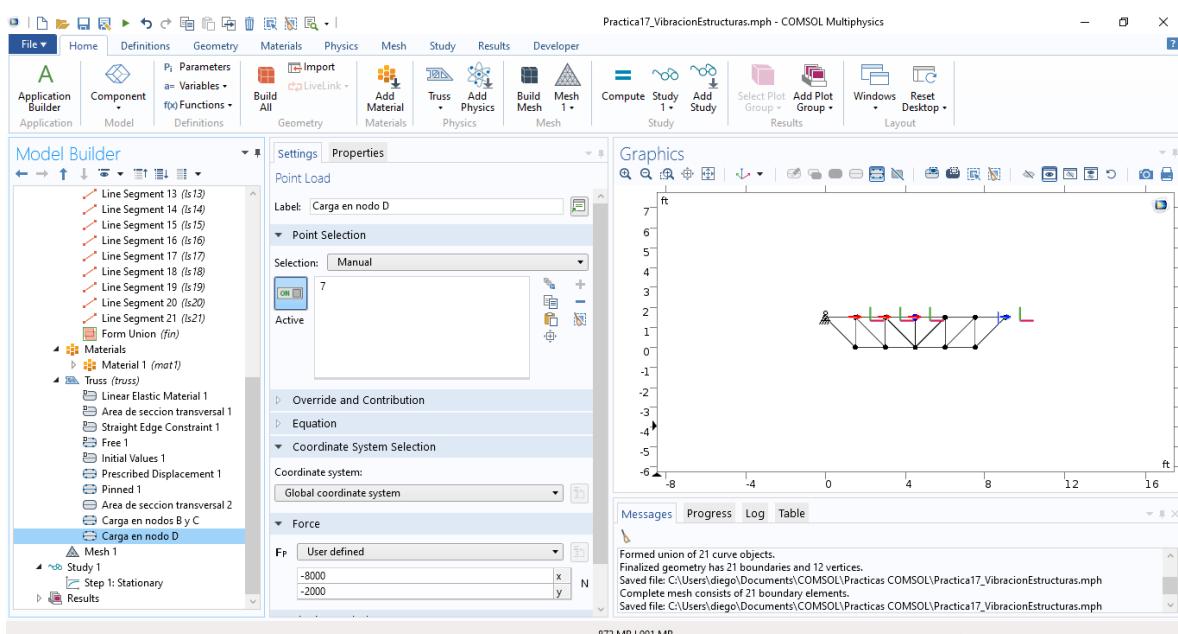
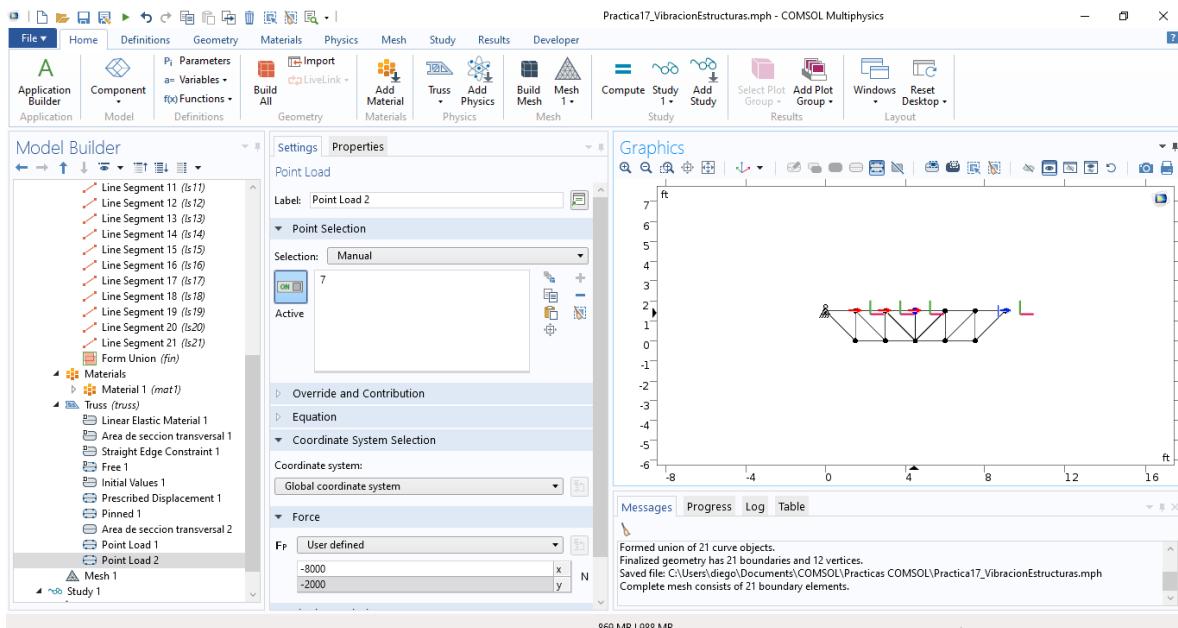


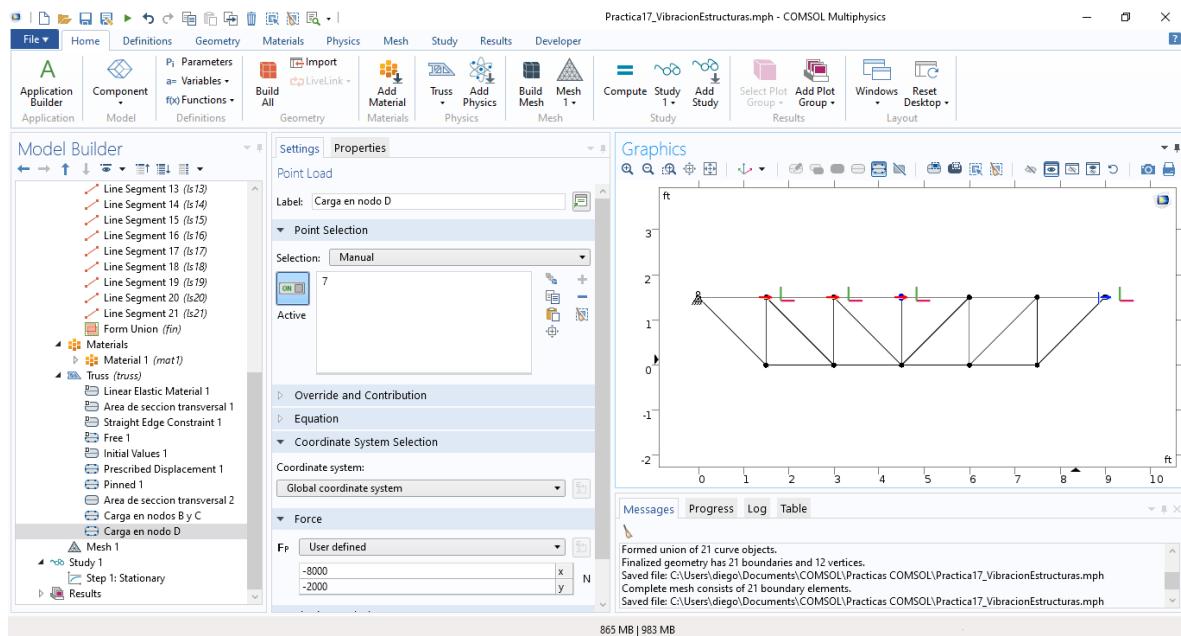




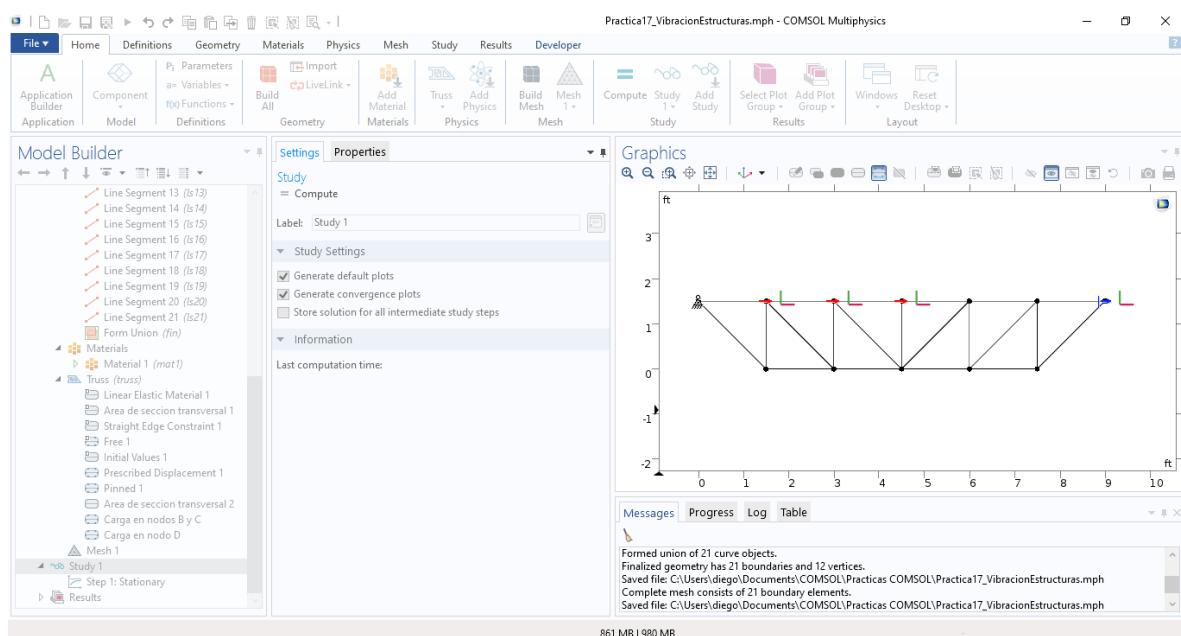
Como las cargas en estos nodos son iguales, las puedo aplicar el mismo Load Point de ambos puntos.

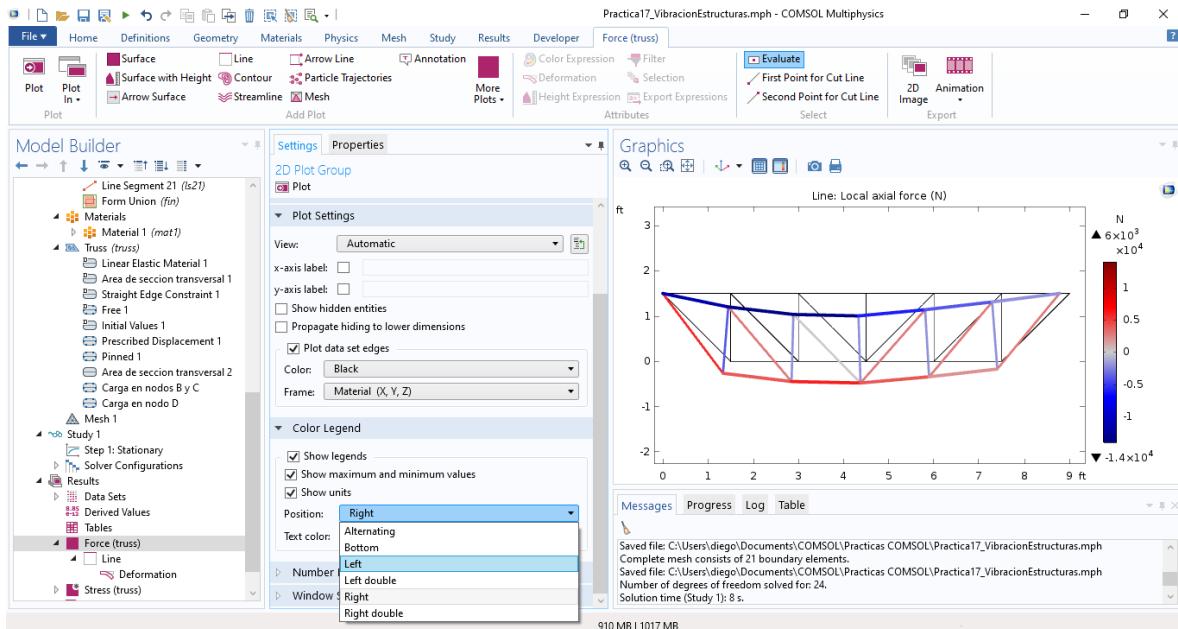
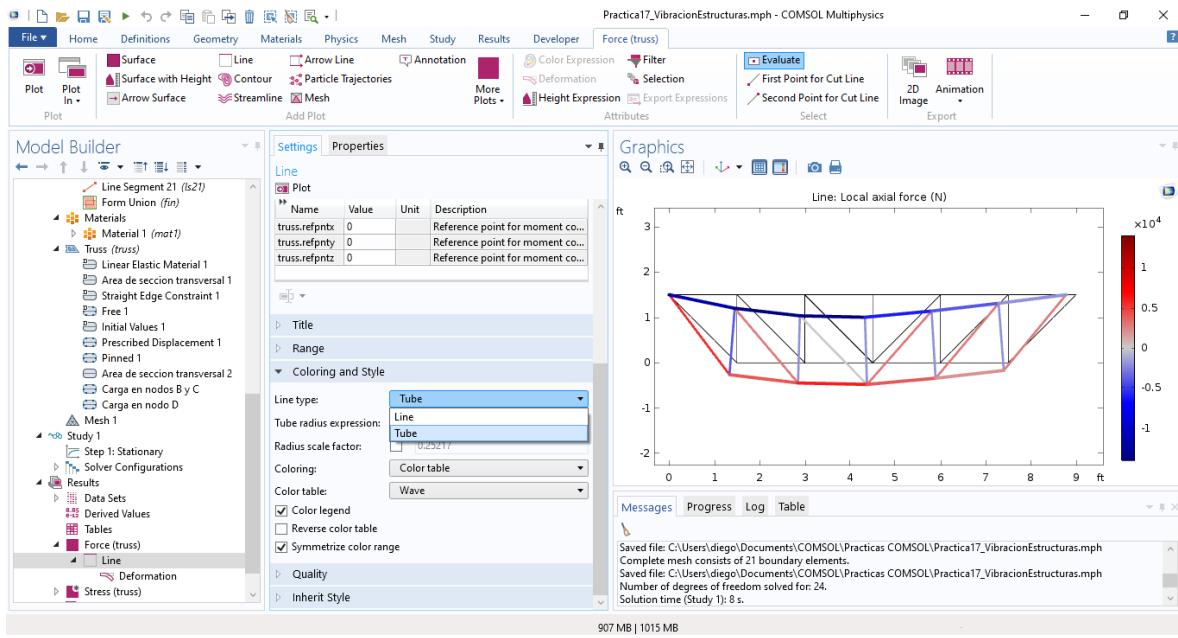


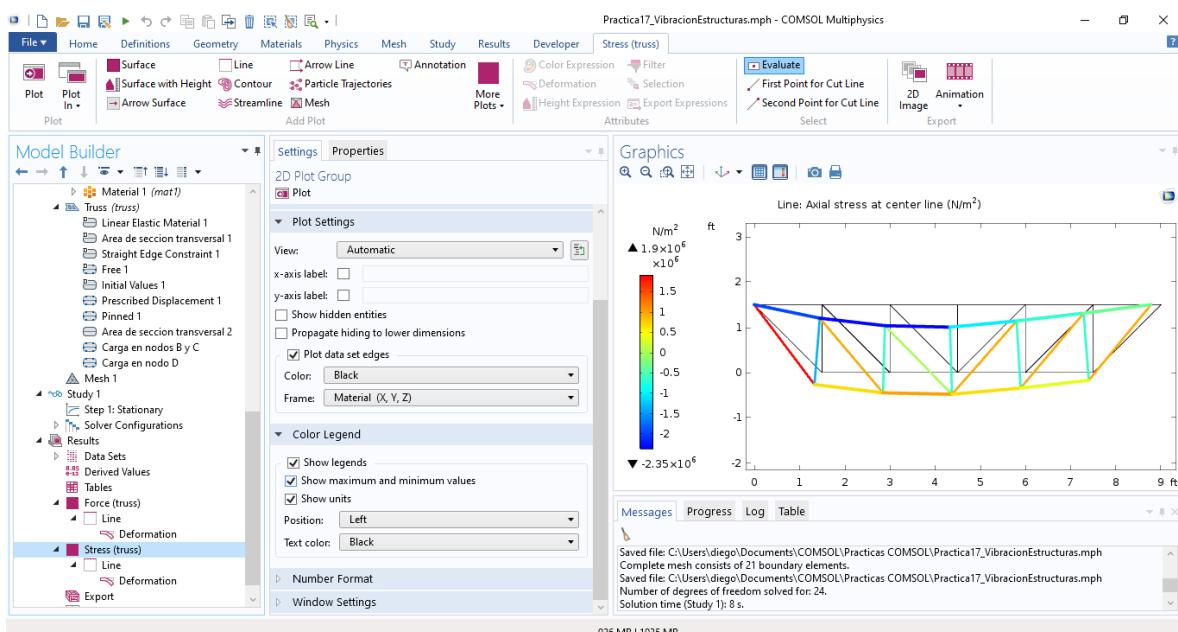
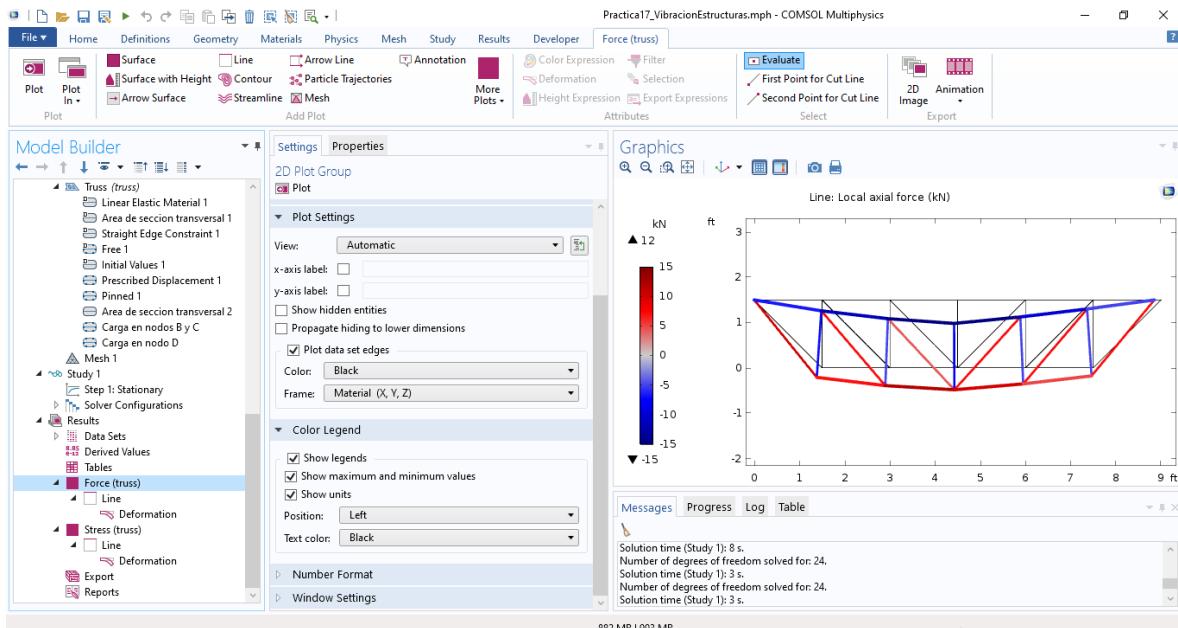


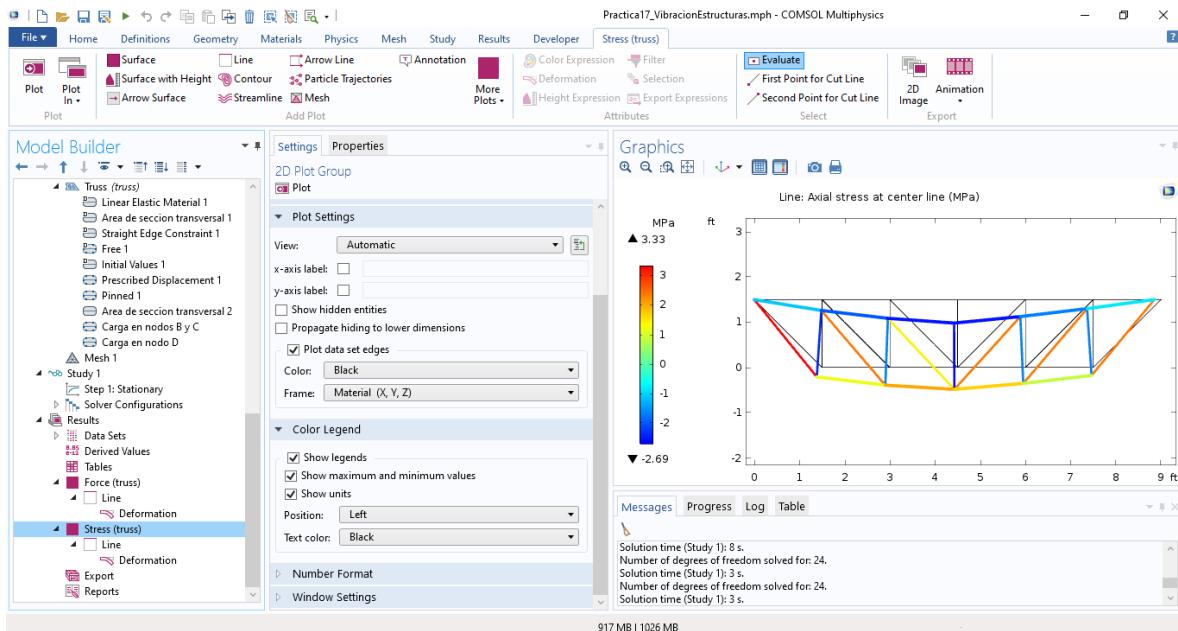


## RESULTADO DEL ELEMENTO FINITO EN COMSOL:

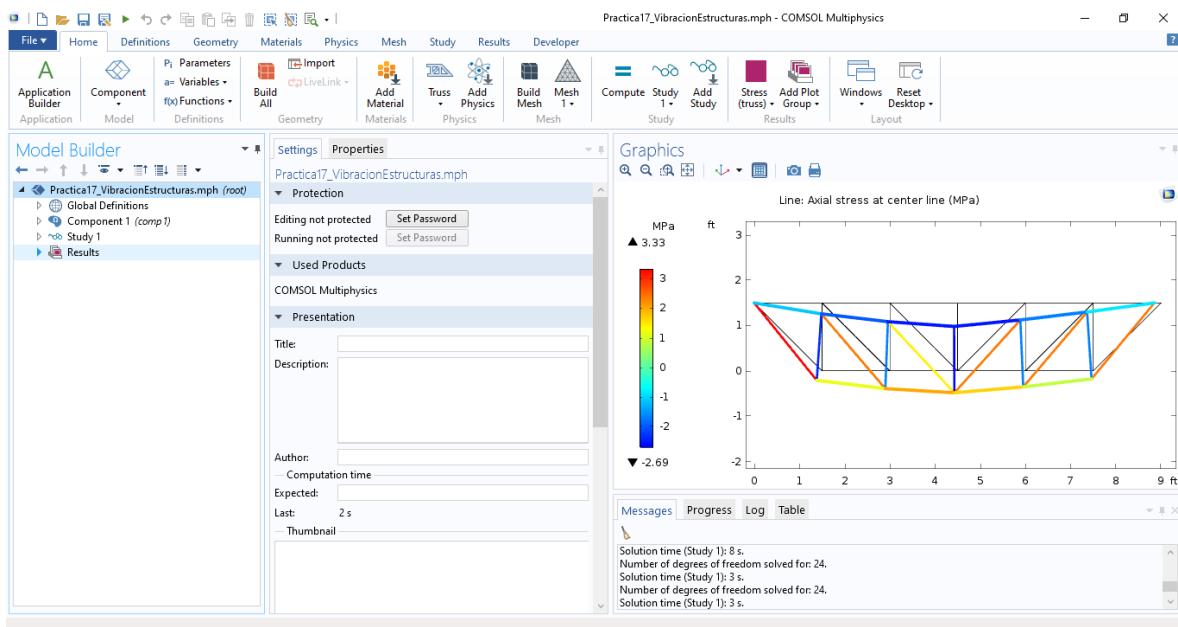


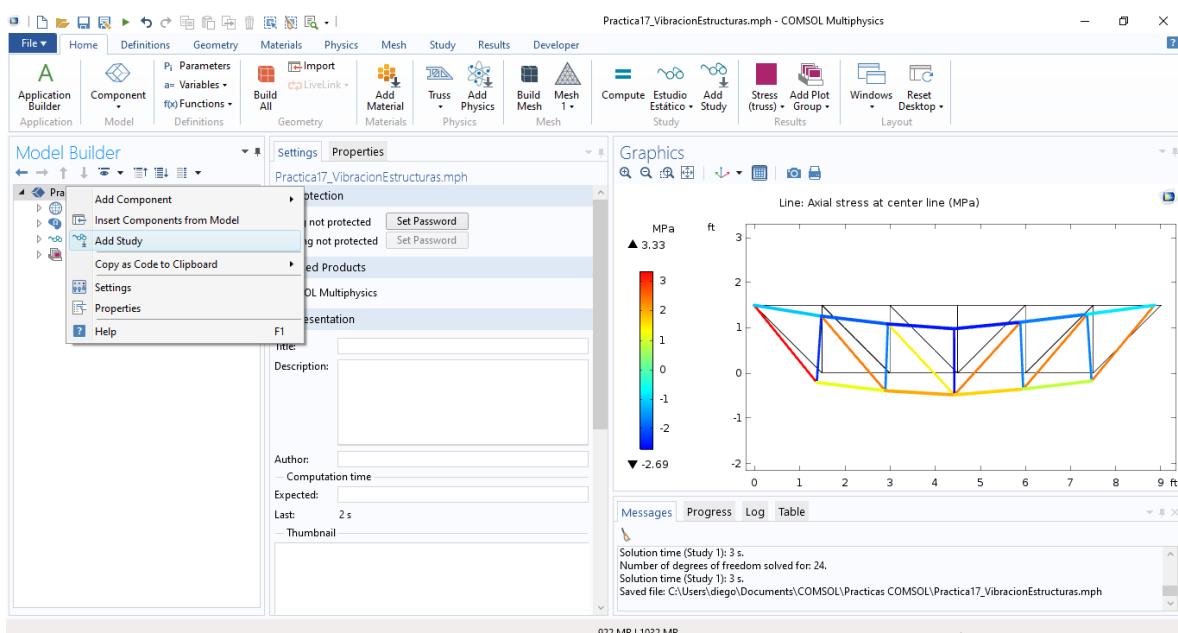
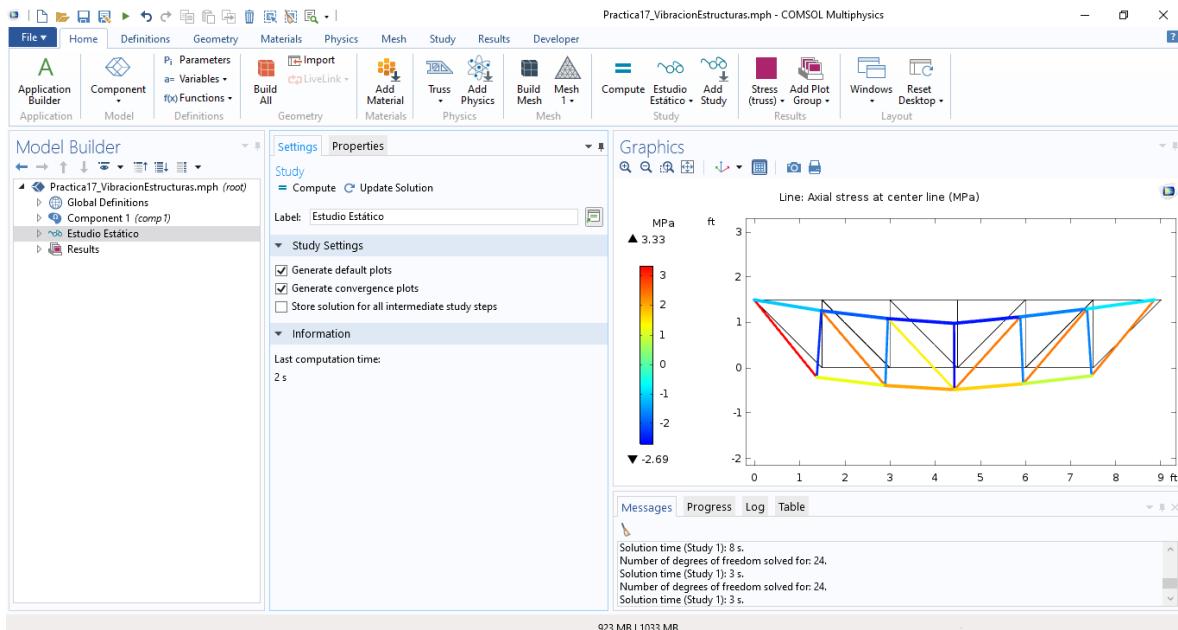


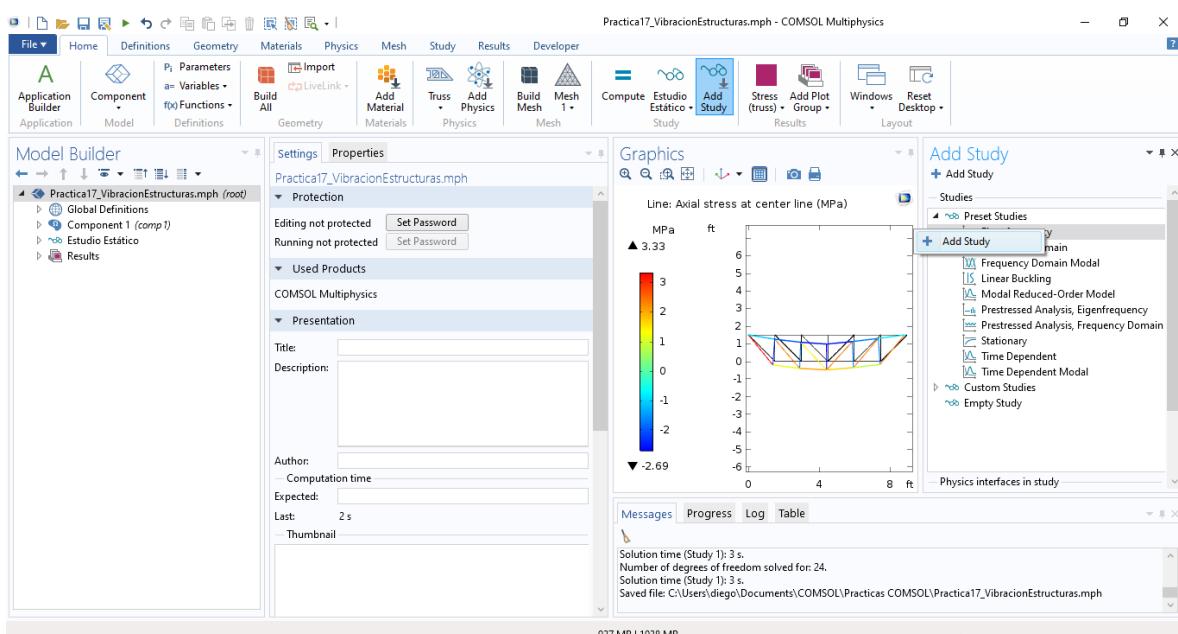
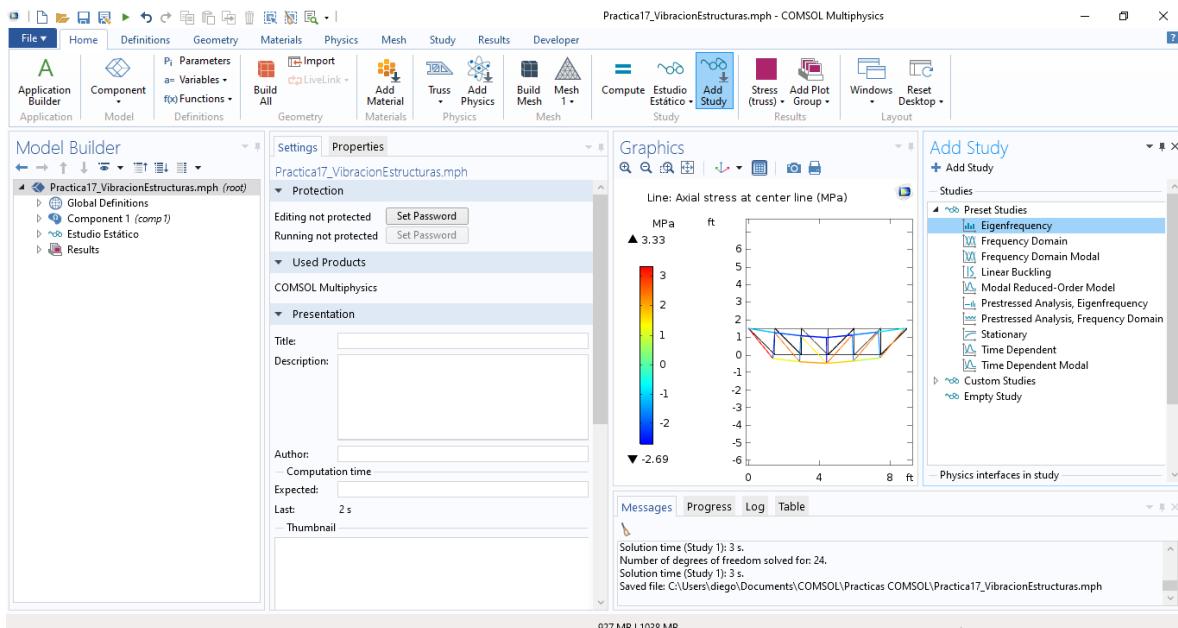


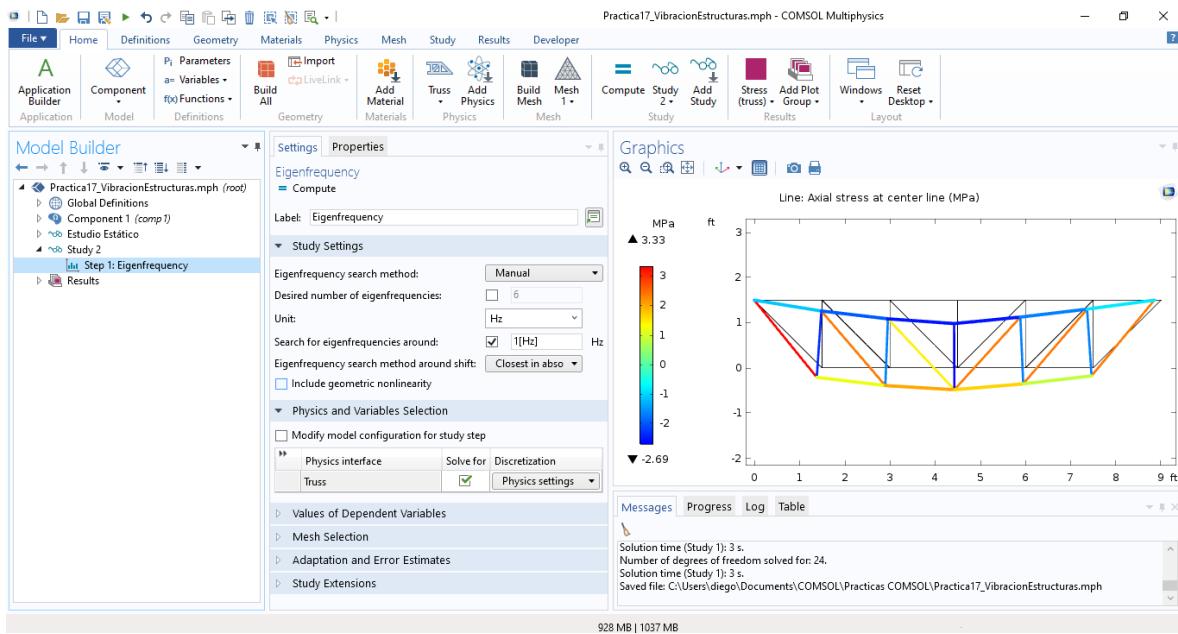


Todas las pestañas de la izquierda las minimizo para que agregue otro estudio, que será el de las vibraciones armónicas.

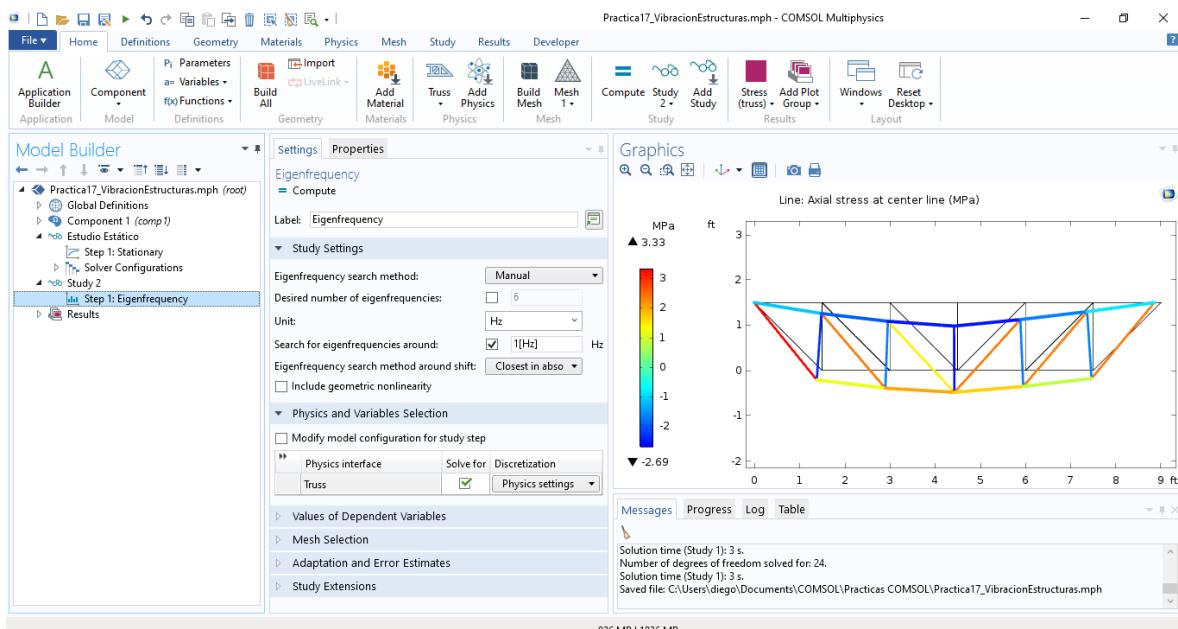


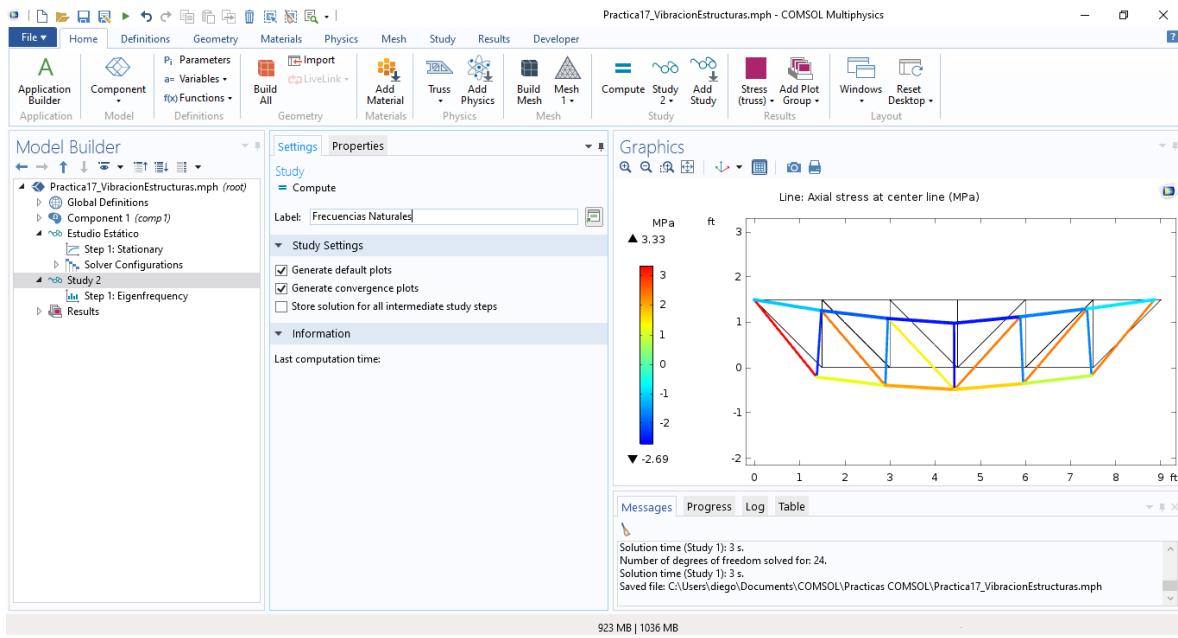




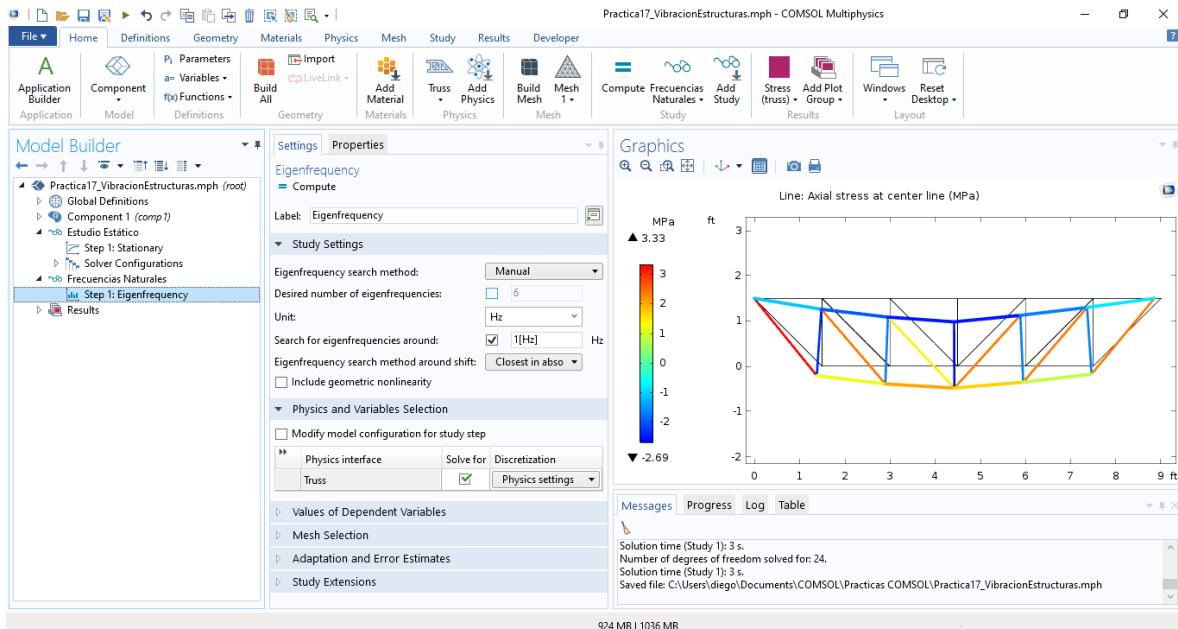


Ya con esto tengo 2 estudios, uno estacionario y otro que será el que medirá la frecuencia natural de la estructura y será transitorio.

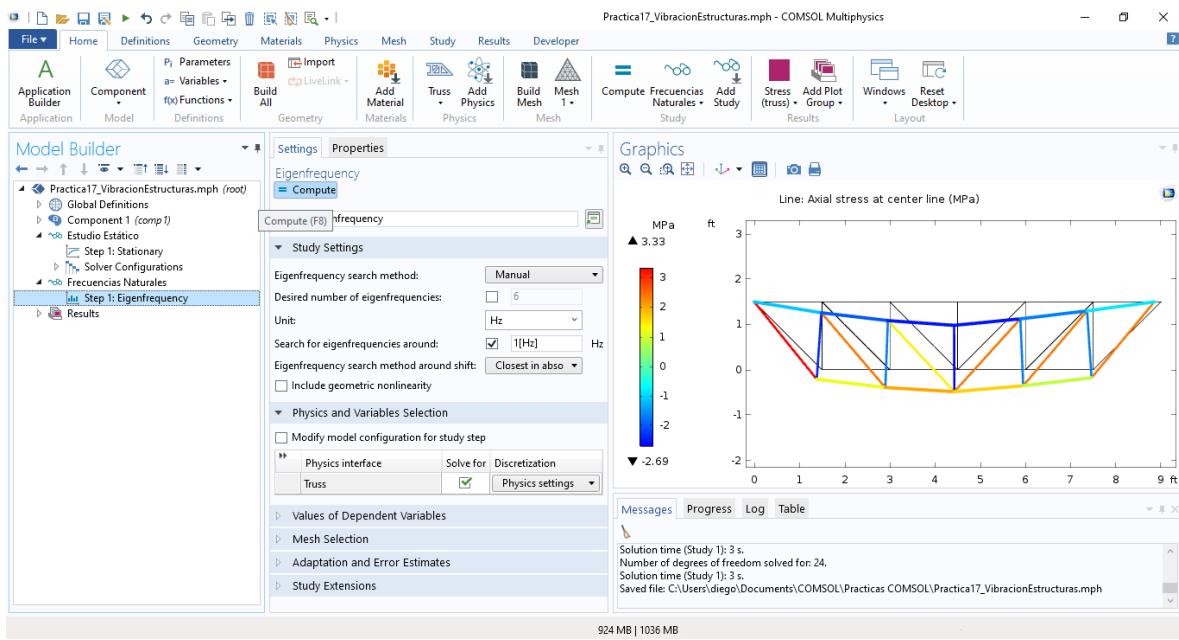




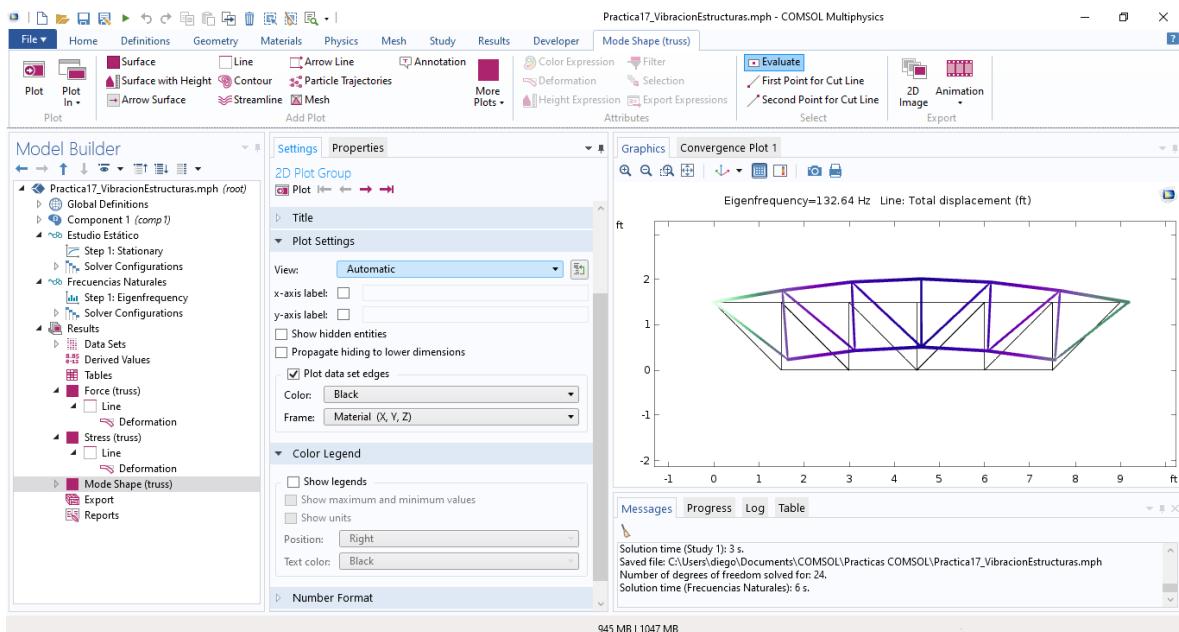
A este estudio de las propiedades de las vibraciones del sistema no le importa las cargas que se estén aplicando, las frecuencias naturales es una propiedad del sistema, así como su masa, su volumen, etc.

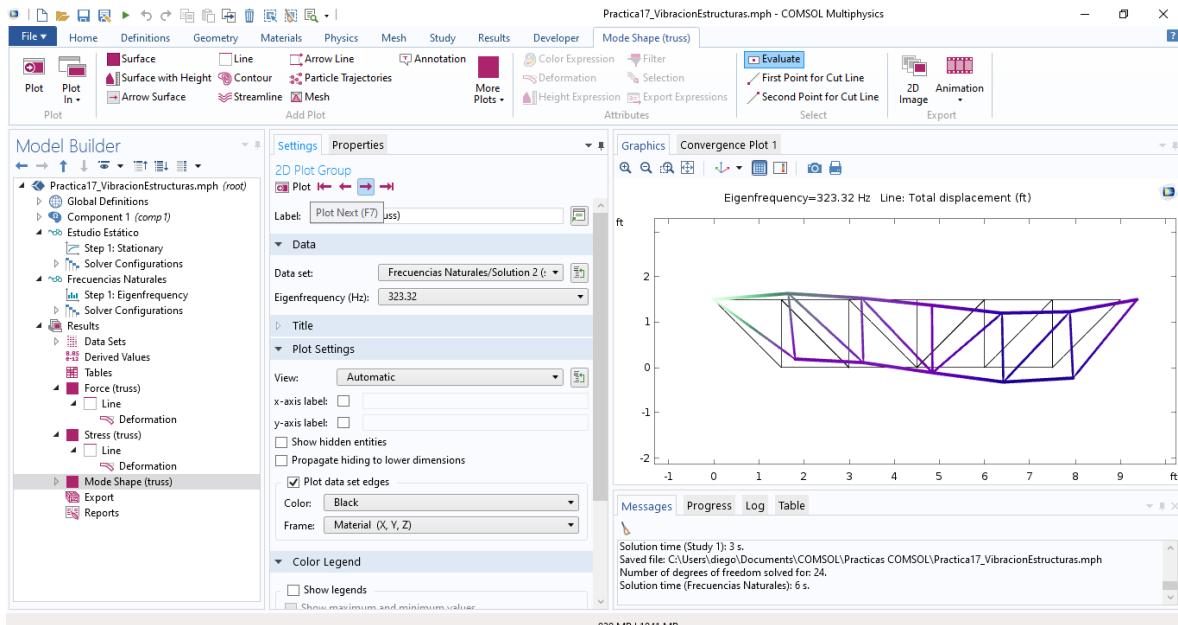
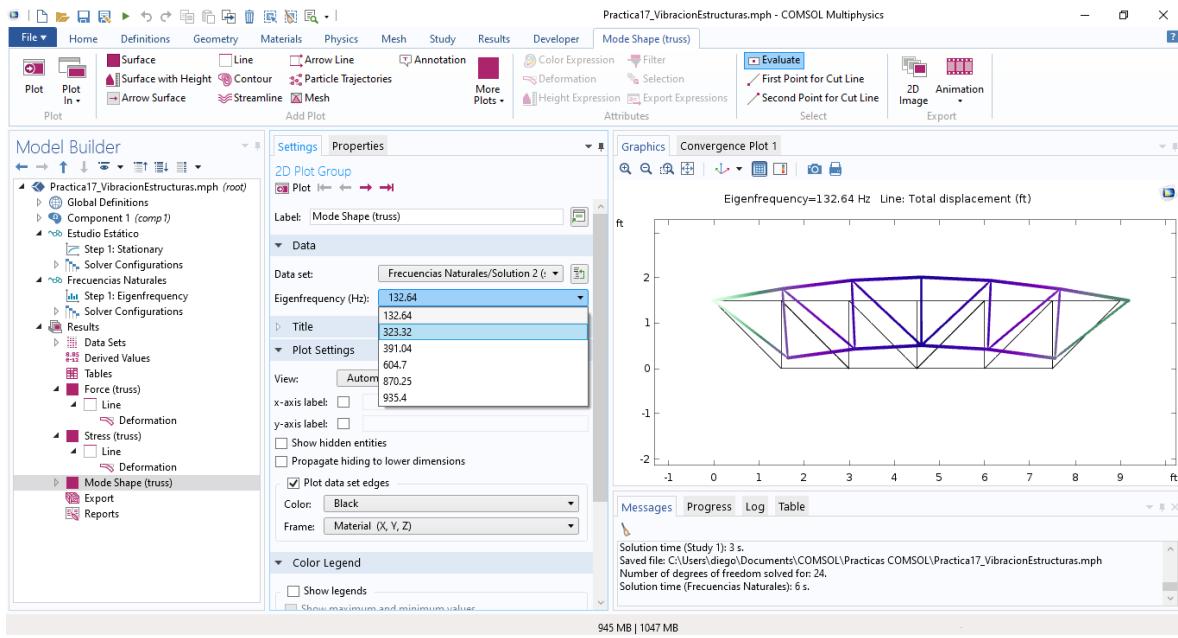


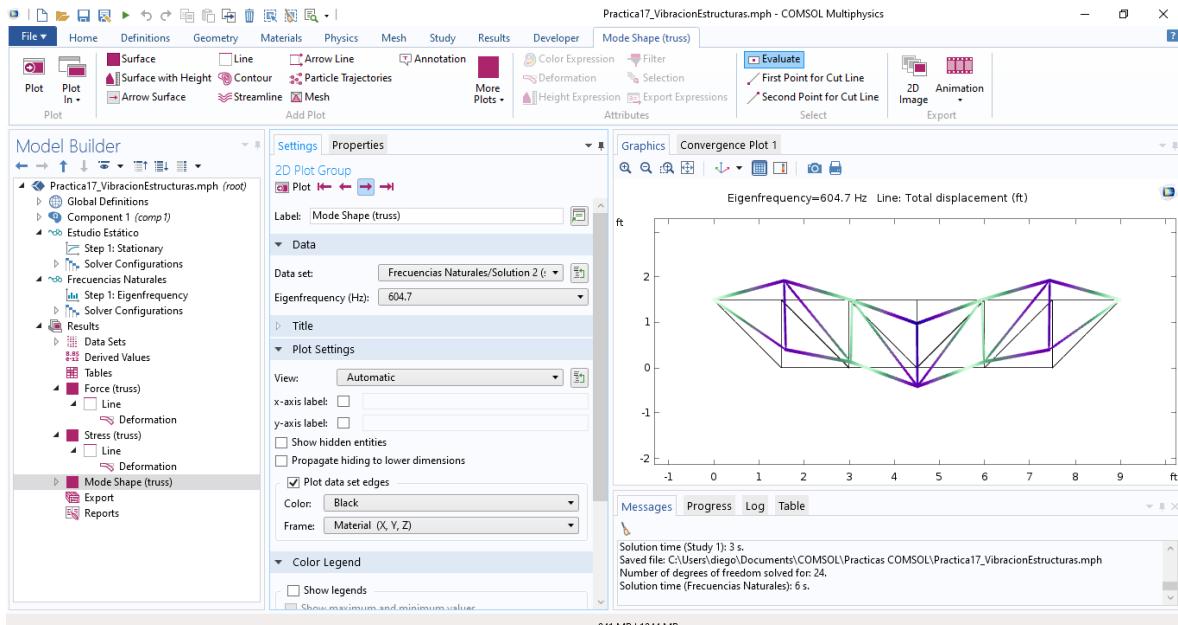
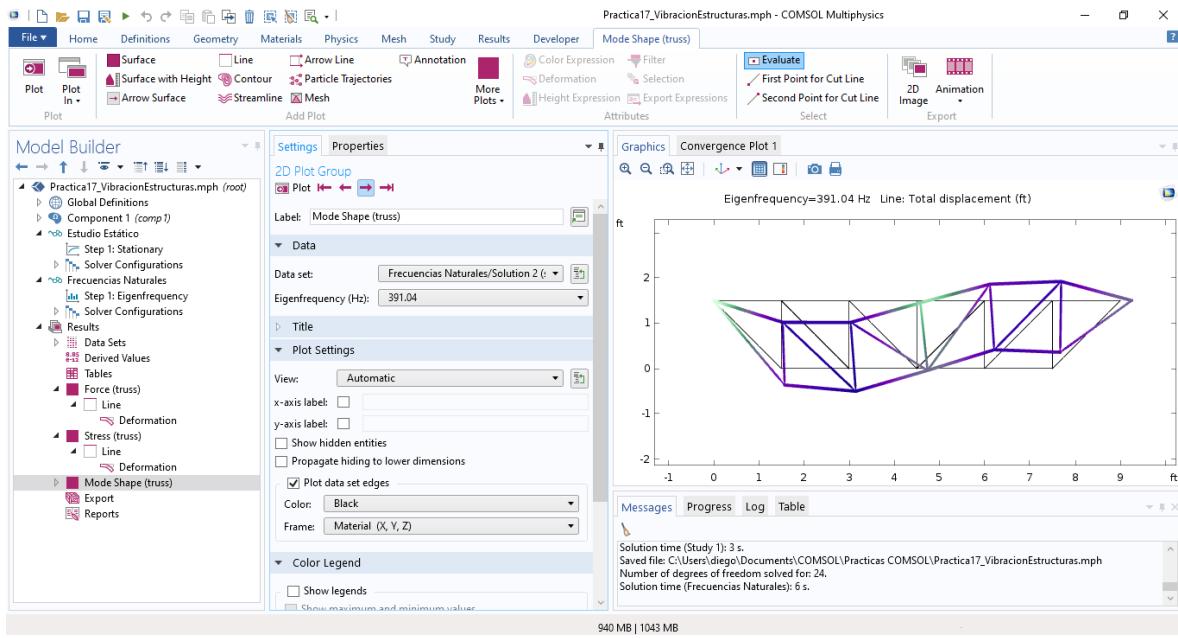
Por default el programa calcula las primeras 6 frecuencias naturales, ya que las más importantes son las primeras.

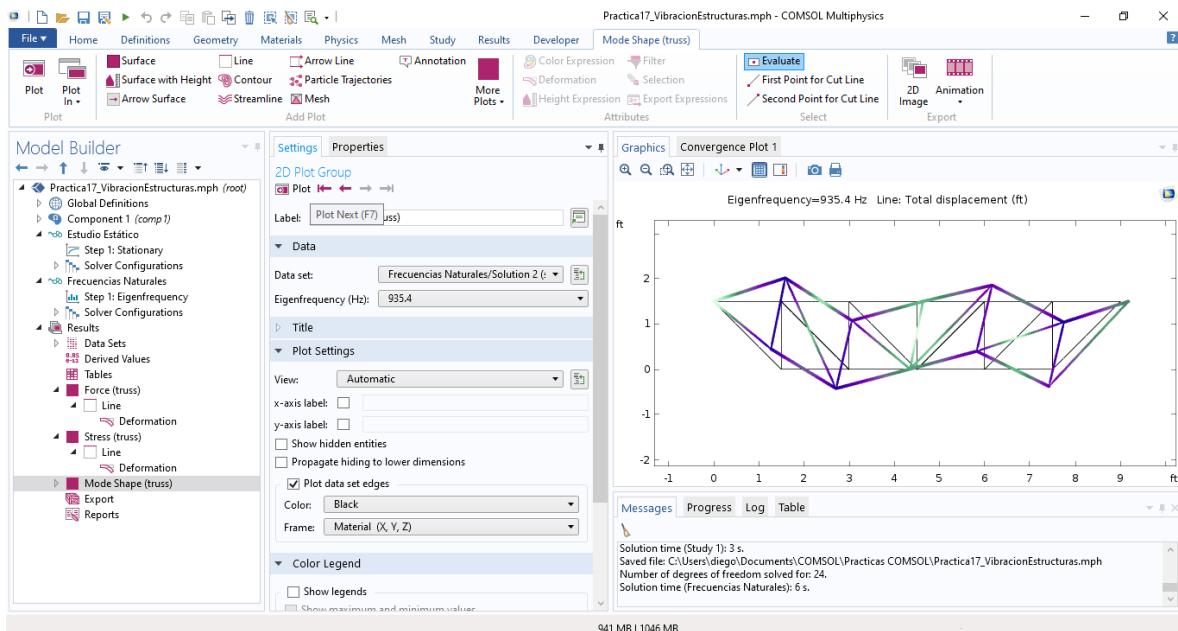
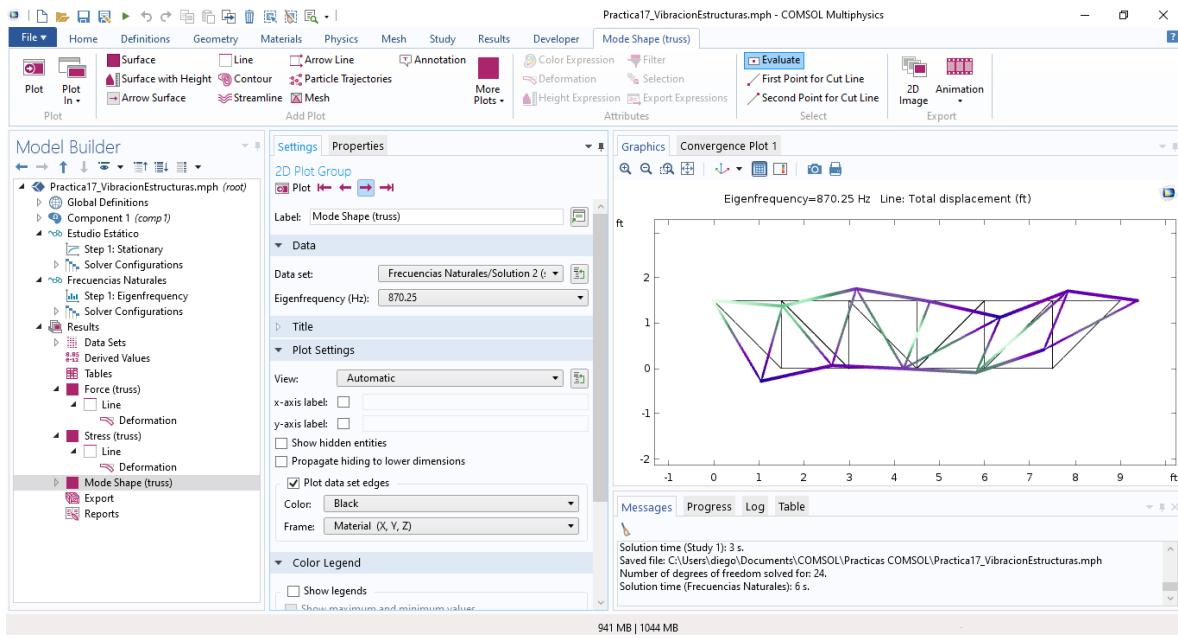


## ANÁLISIS DE VIBRACIONES ARMÓNICAS EN COMSOL:

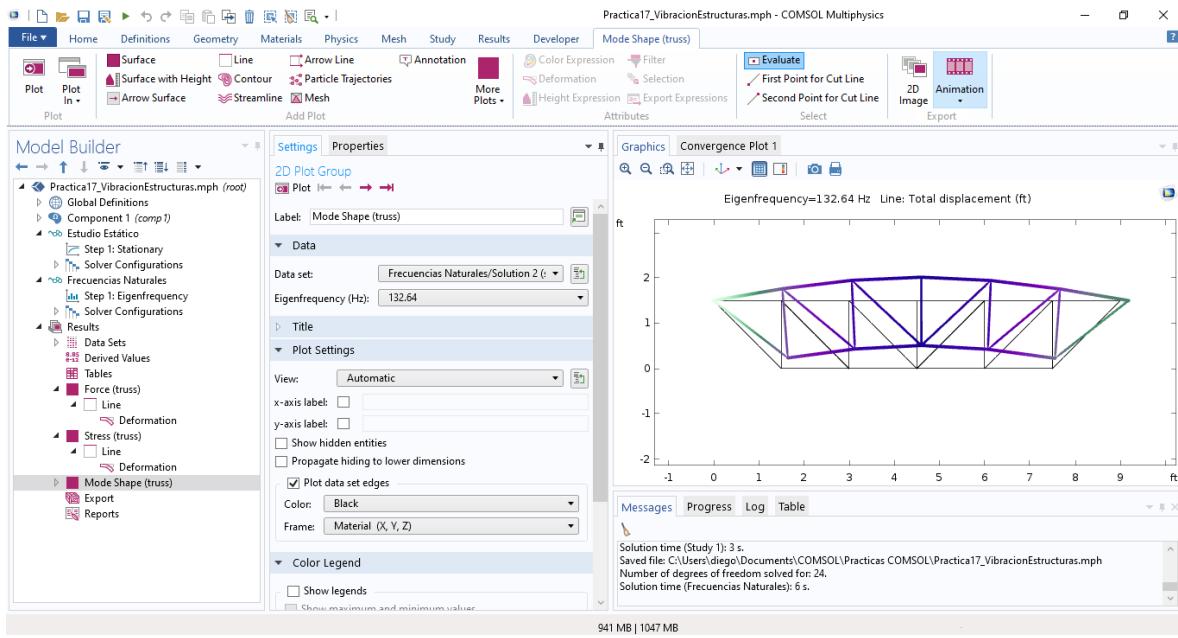




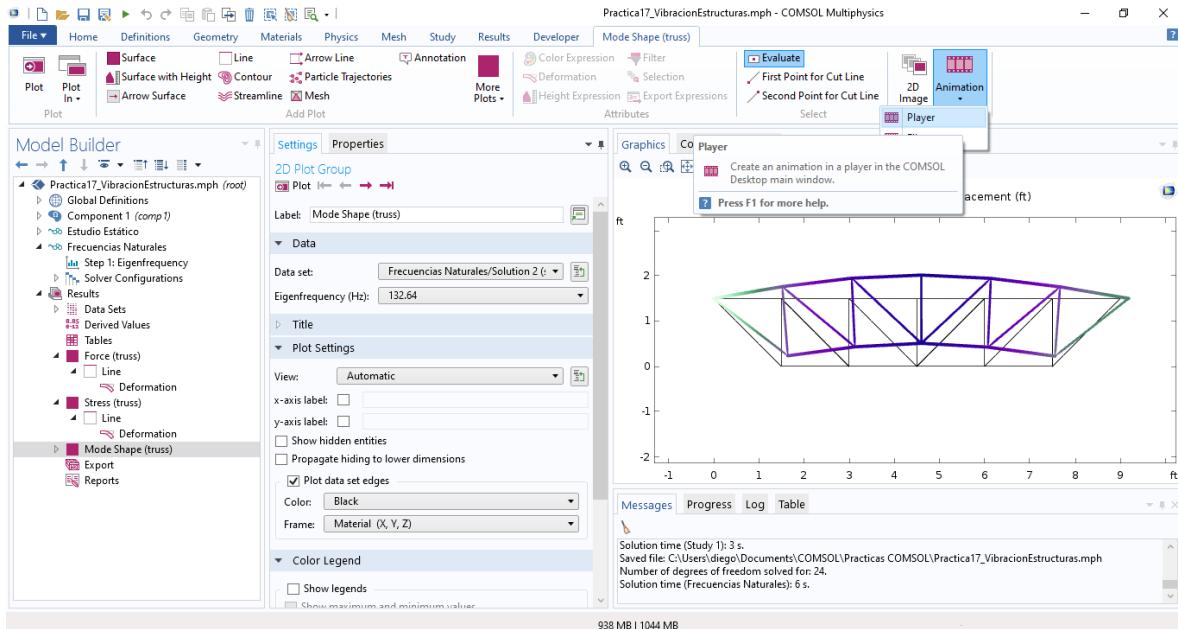


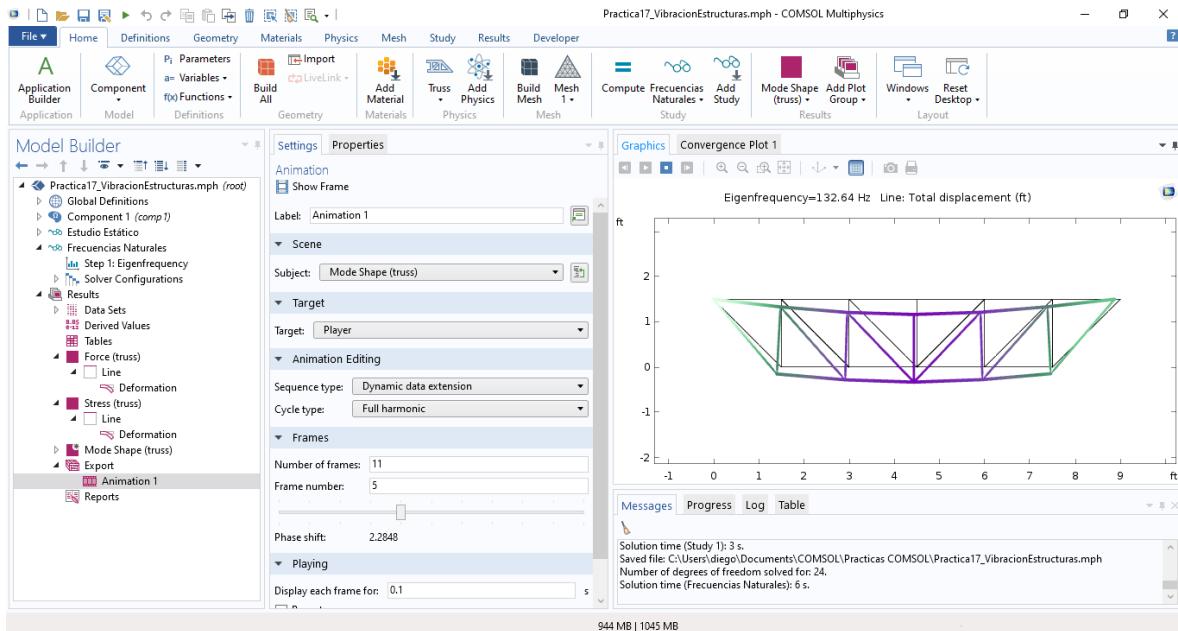
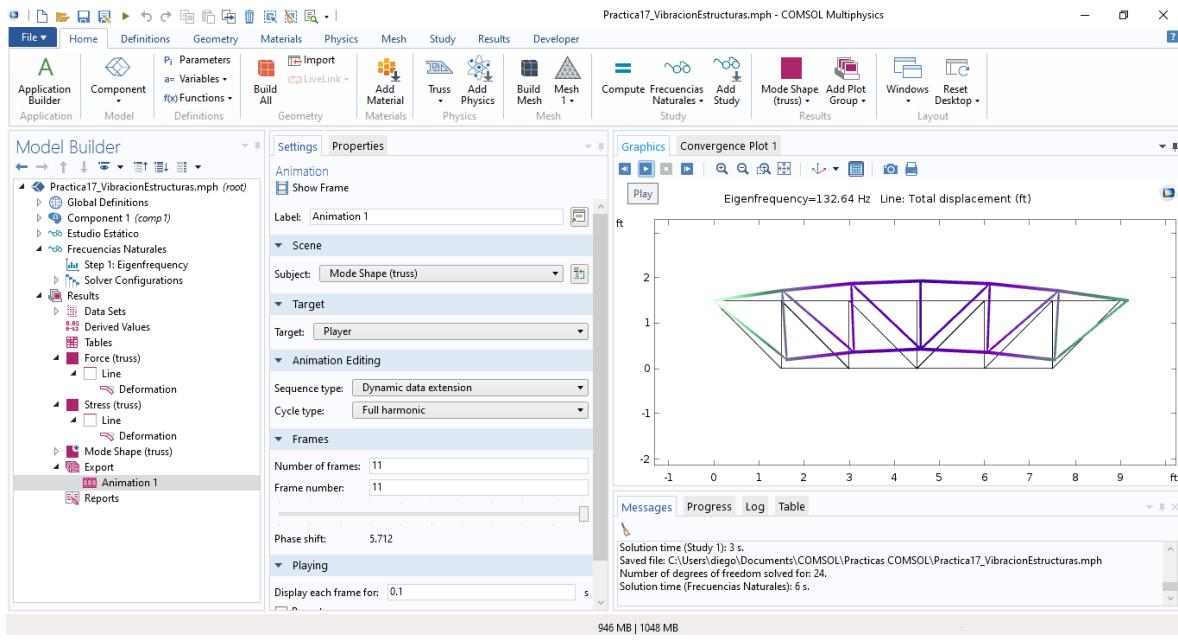


Se pueden observar animaciones del comportamiento de todas las vibraciones encontradas.



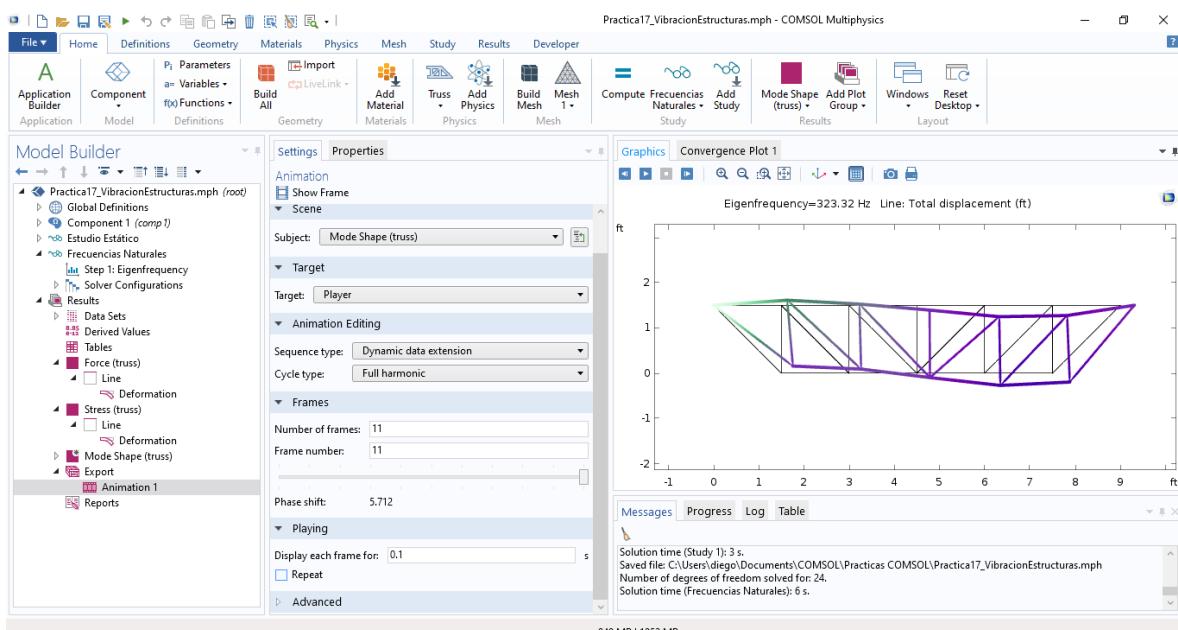
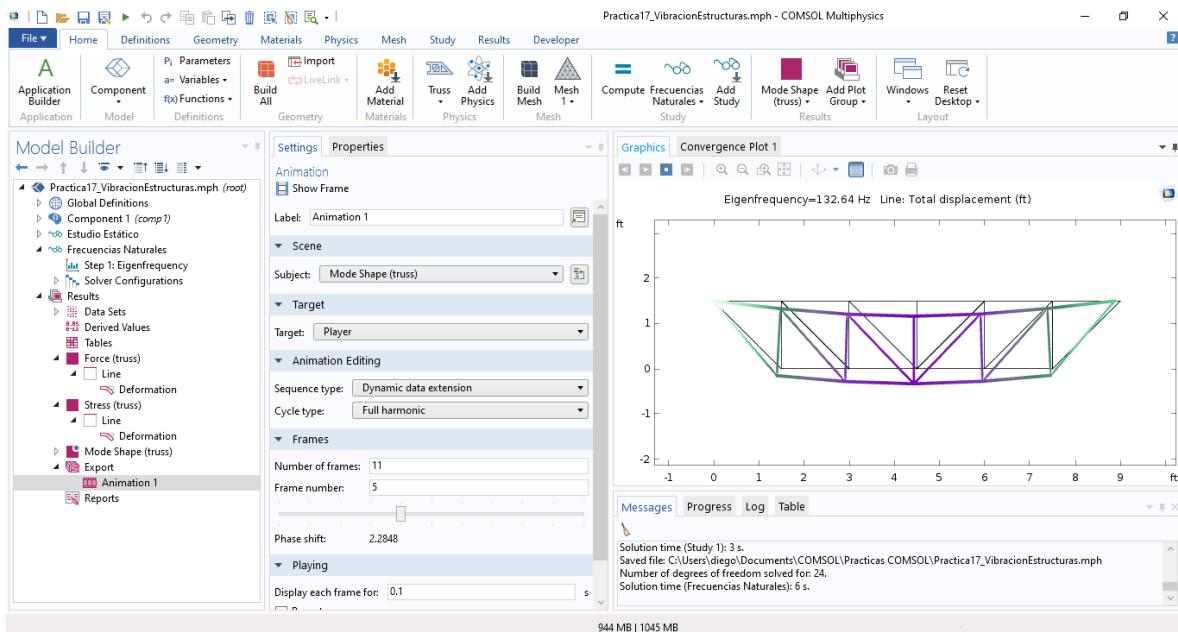
Con la opción de Player solo le doy play a la animación y con la opción de File se guarda la animación.

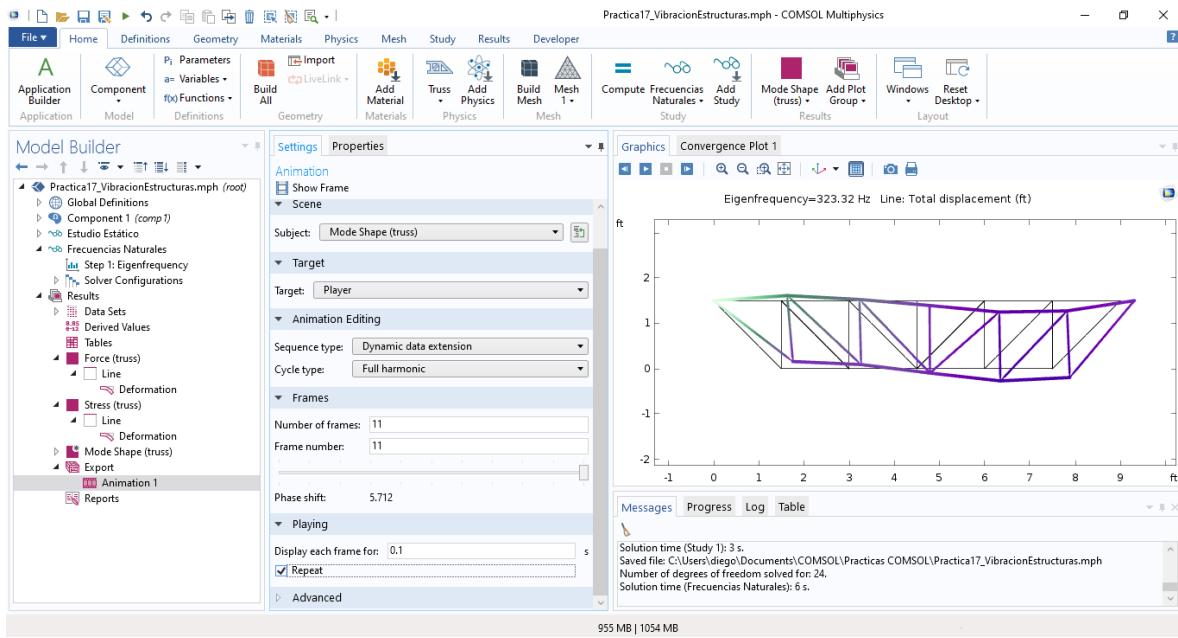




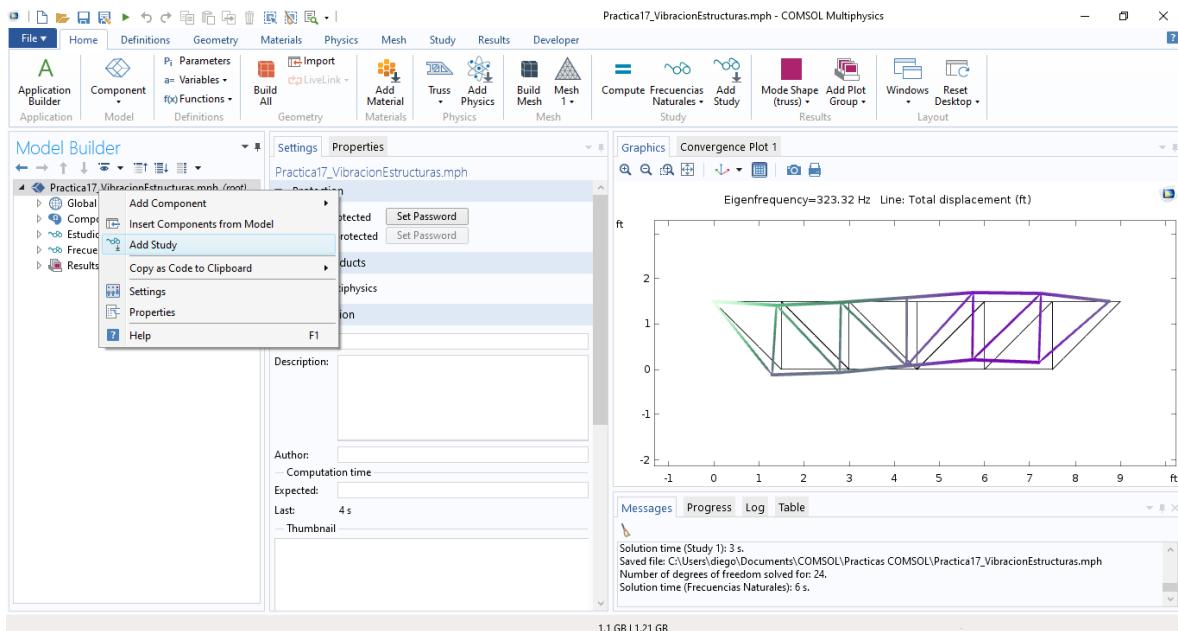
Dentro de la opción de Mode Shape es donde elijo la frecuencia que quiero ver.

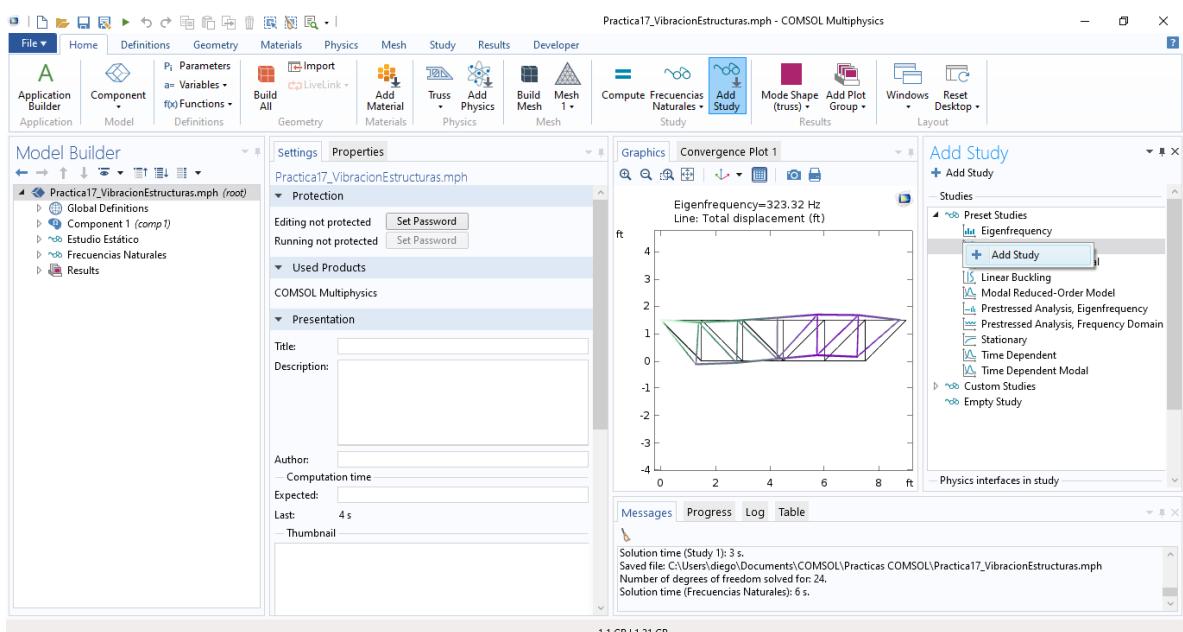
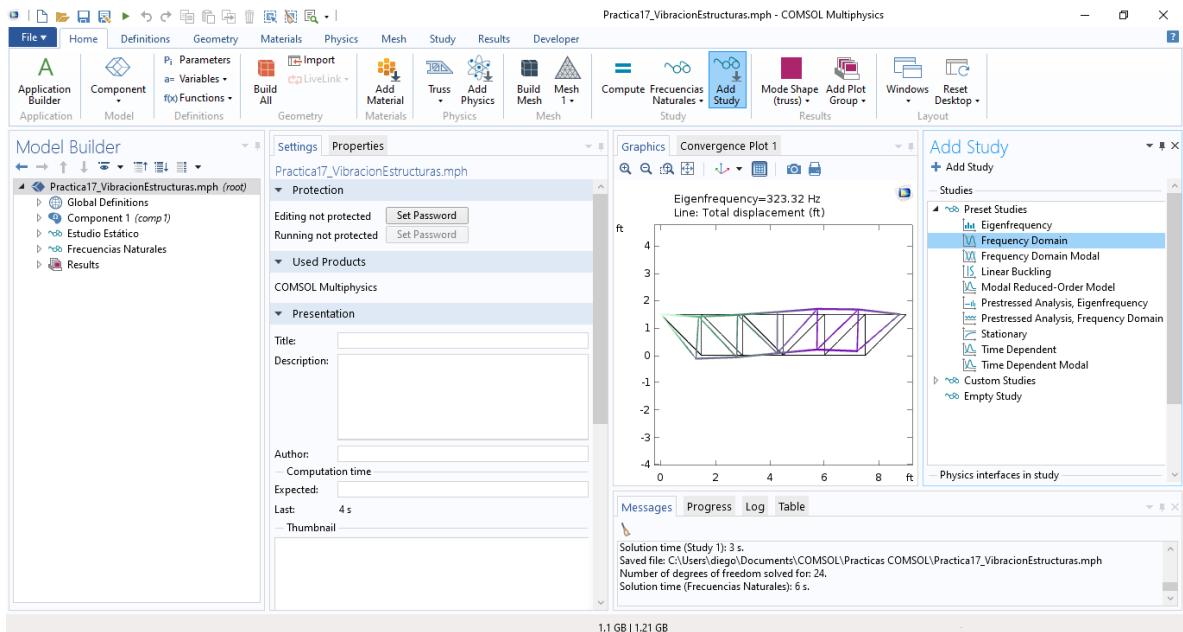


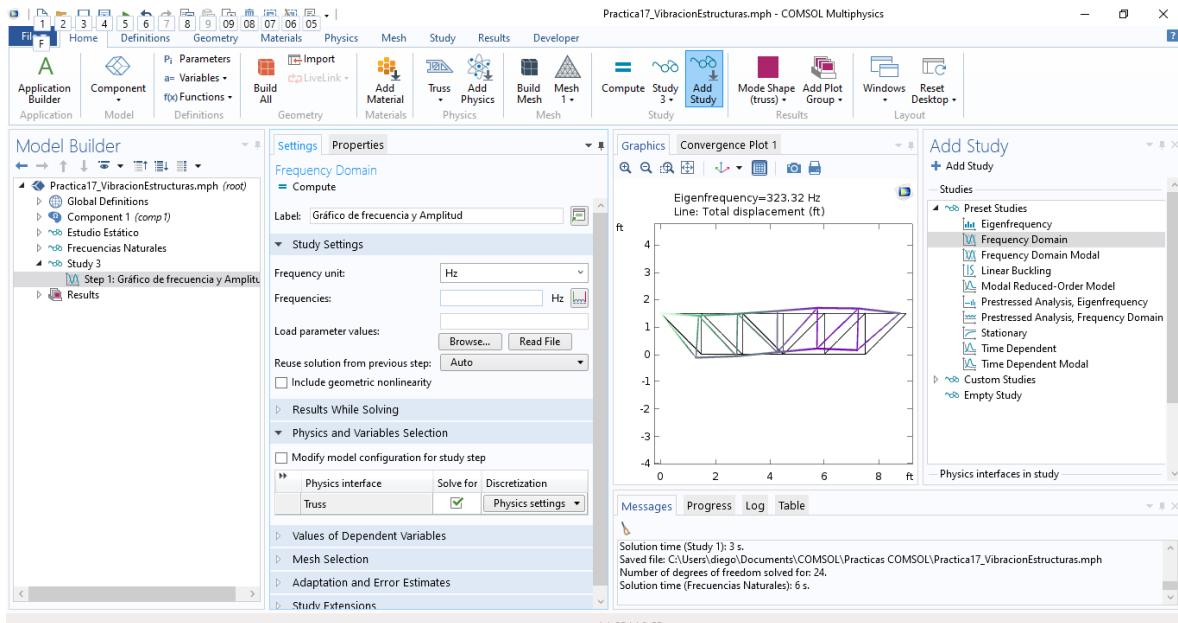
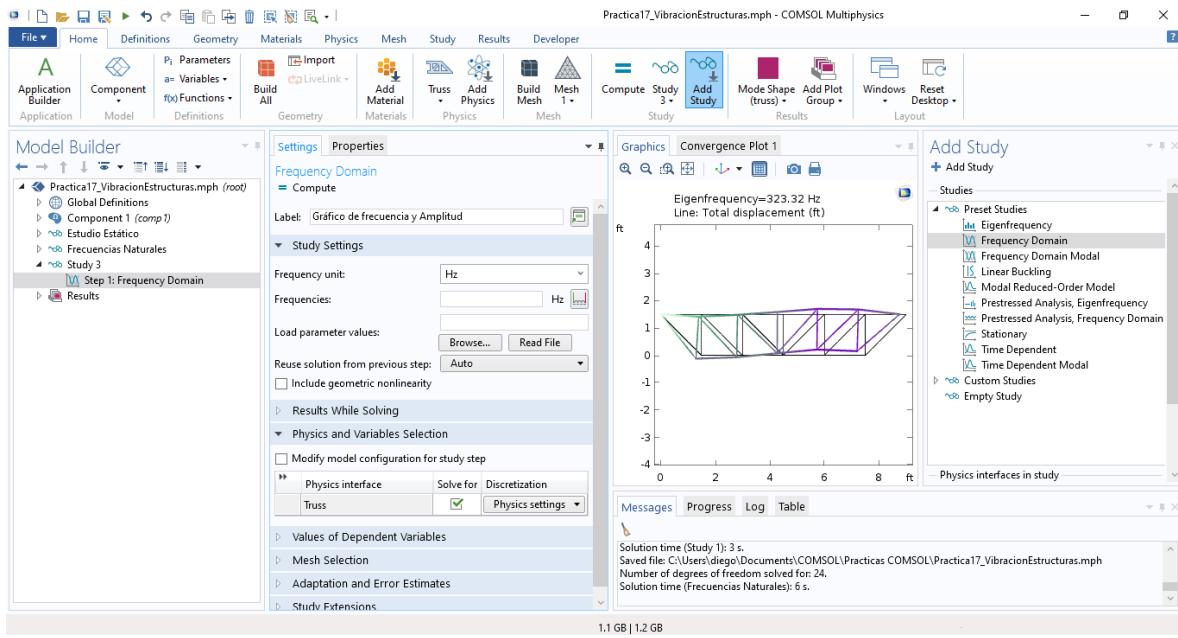




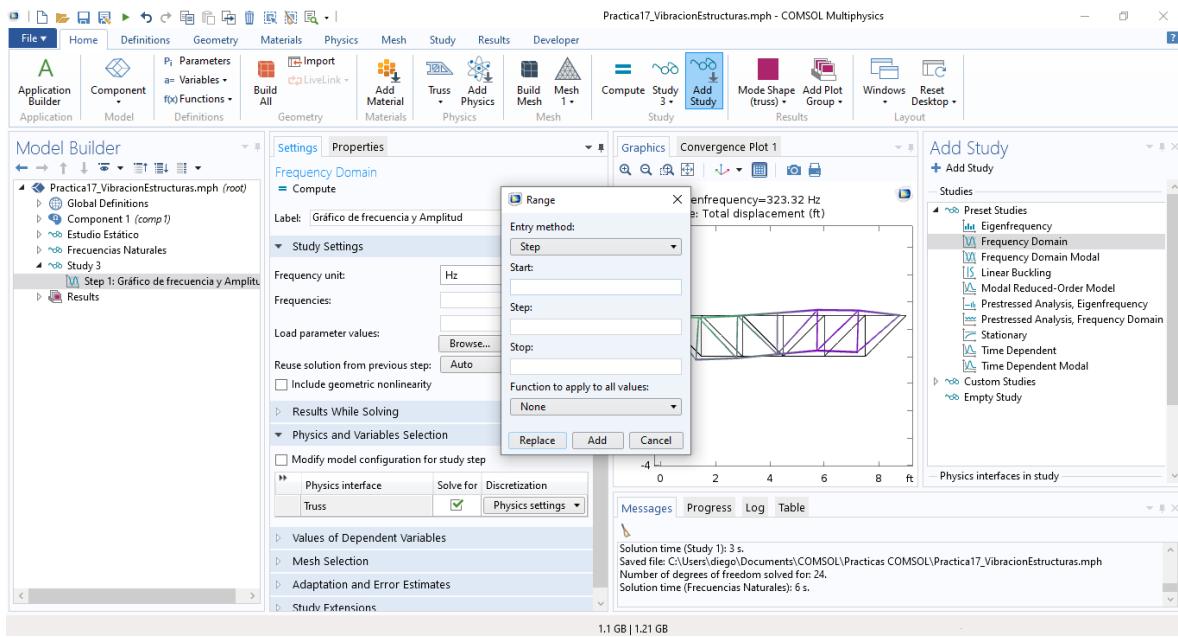
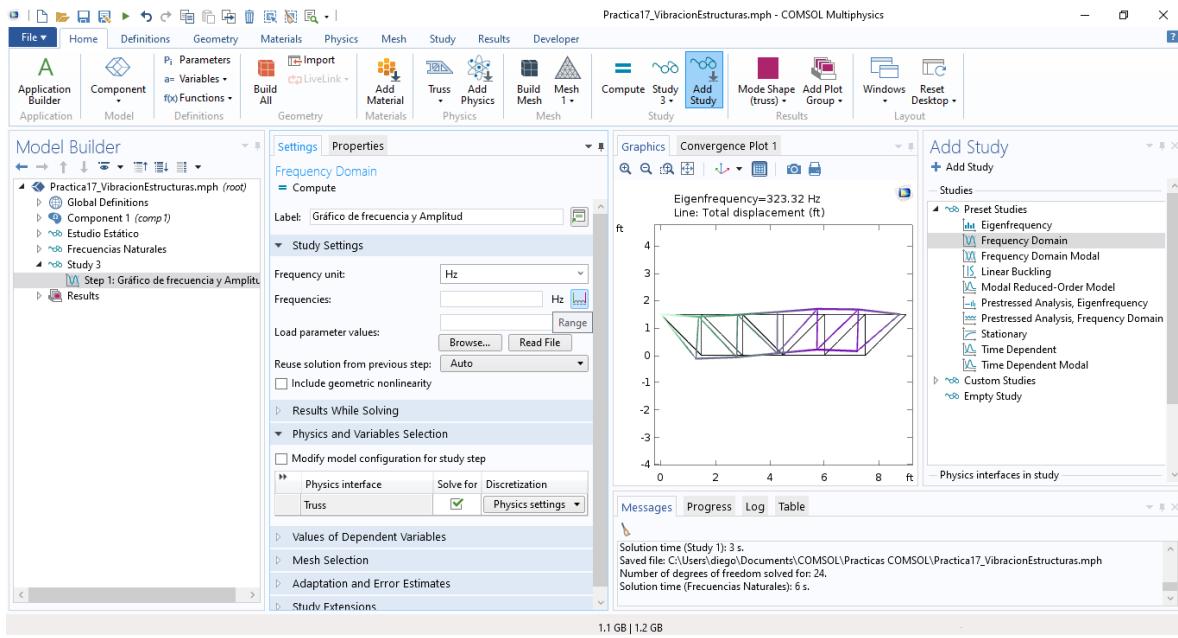
Toda la animación se puede poner dentro de una gráfica, pero para esto debo agregar otro estudio.



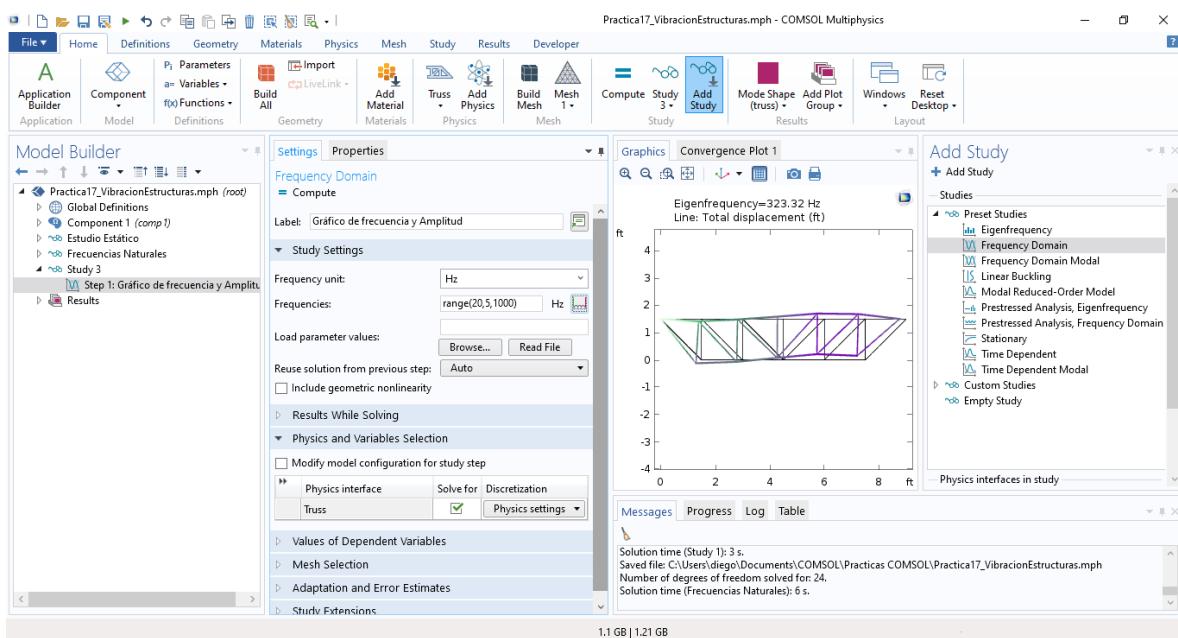
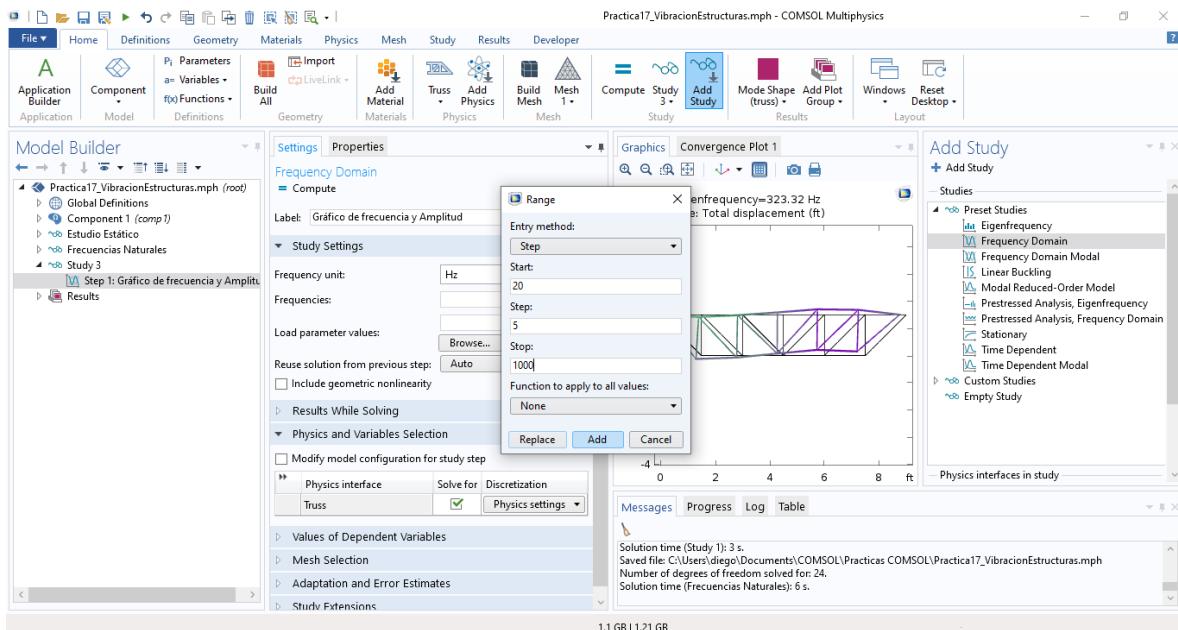


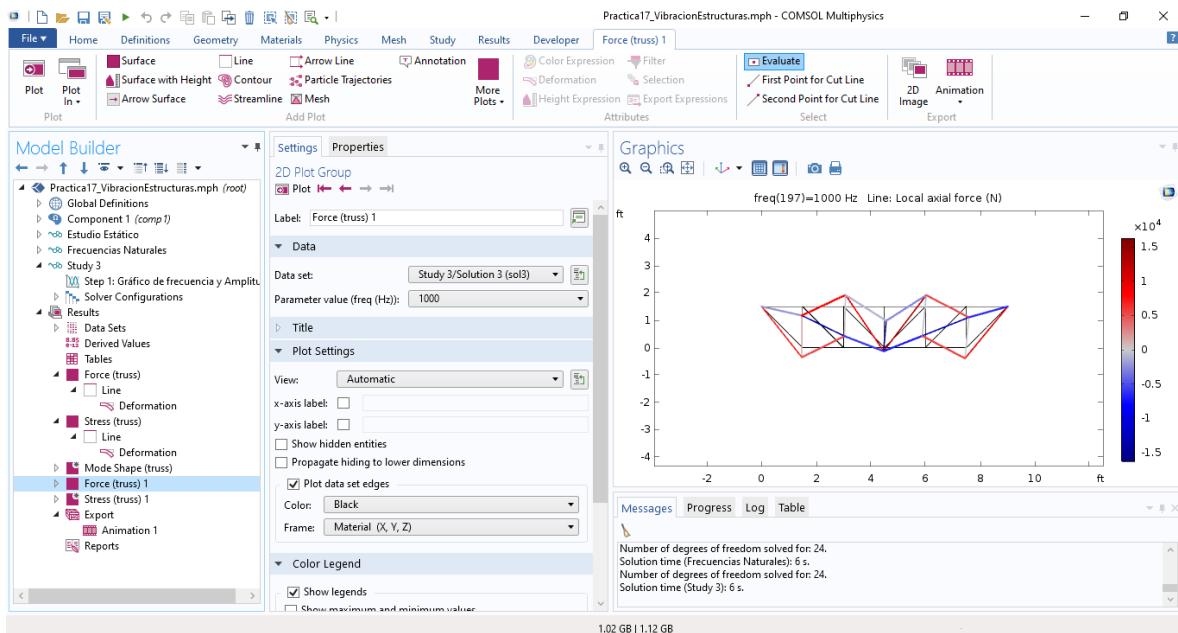
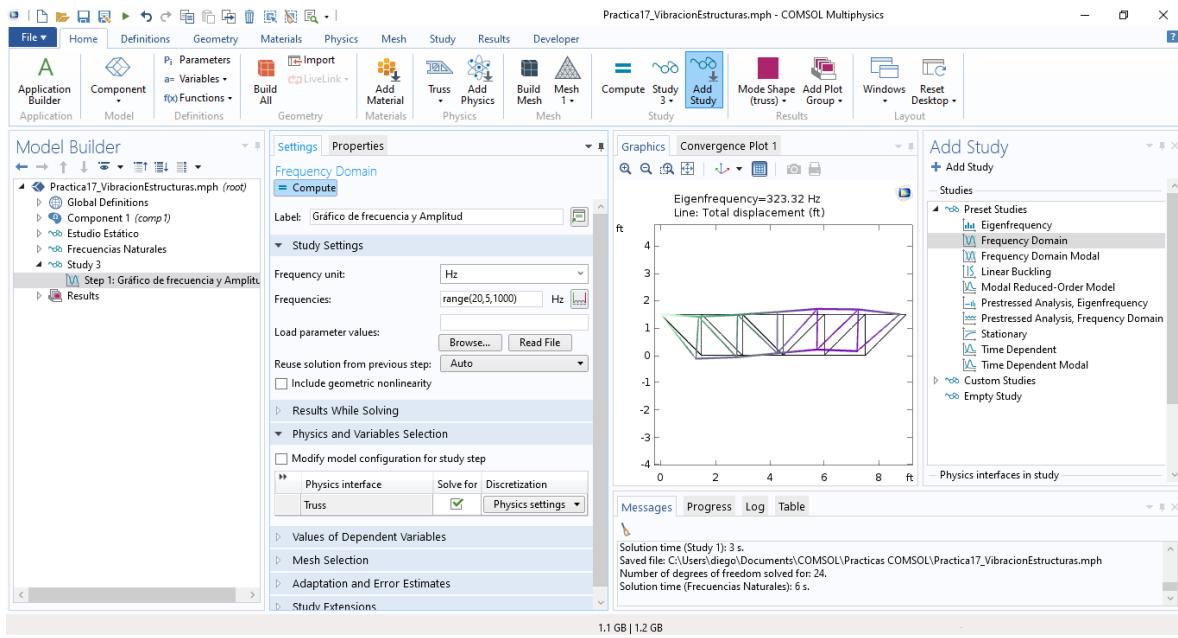


Donde dice frequencies debo meter el rango de las frecuencias que quiero graficar, esto es lo que irá en el eje horizontal de la gráfica resultante.

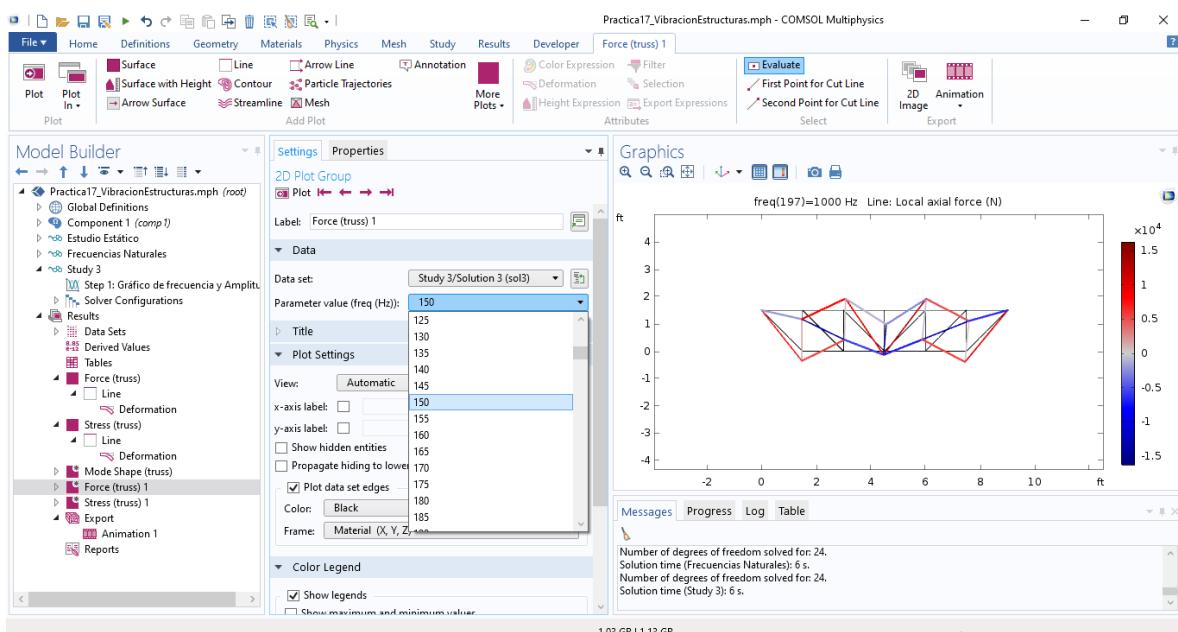
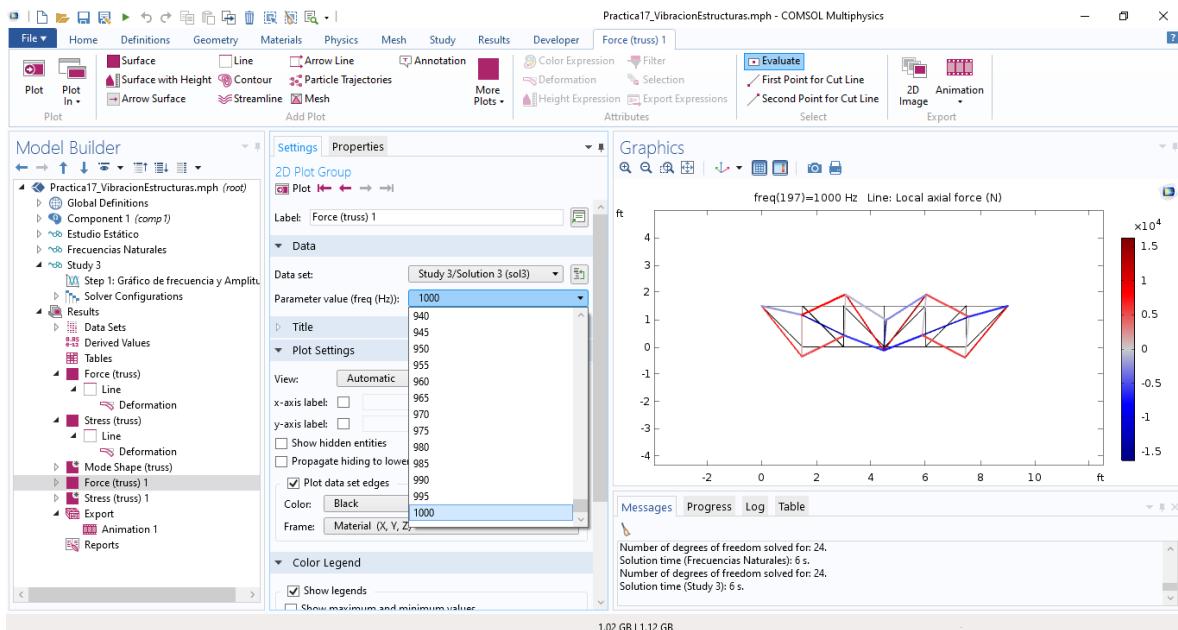


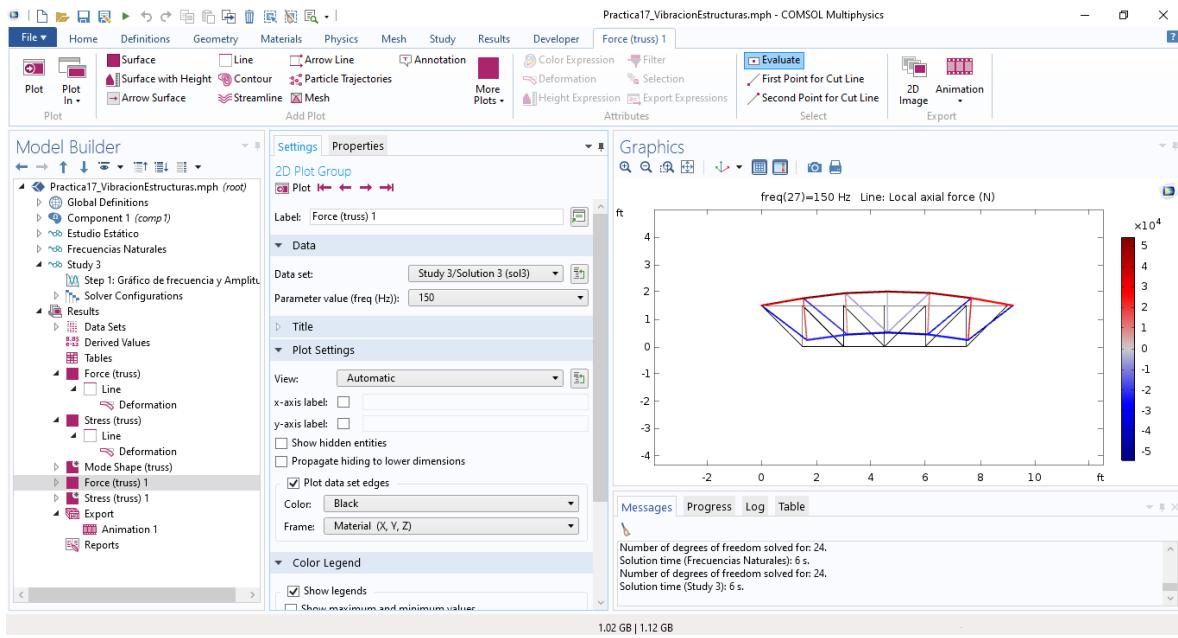
En esta ventana emergente le pongo su valor inicial y final y la separación entre ellos.





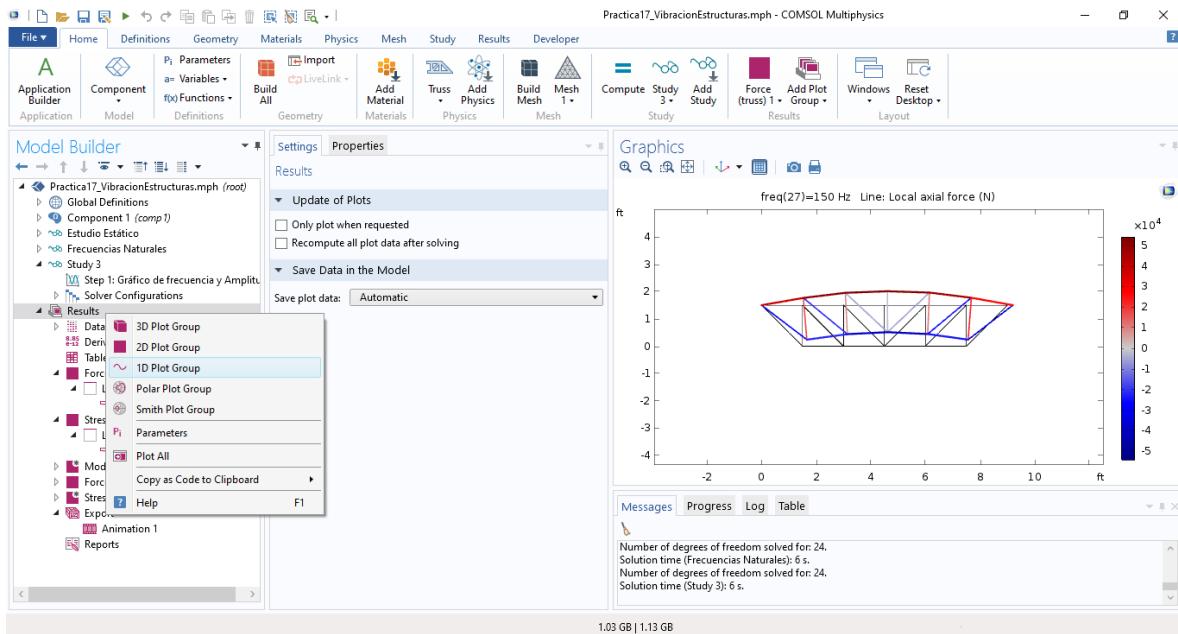
Aquí me mostrará que le pasará para todas las frecuencias que le indique.

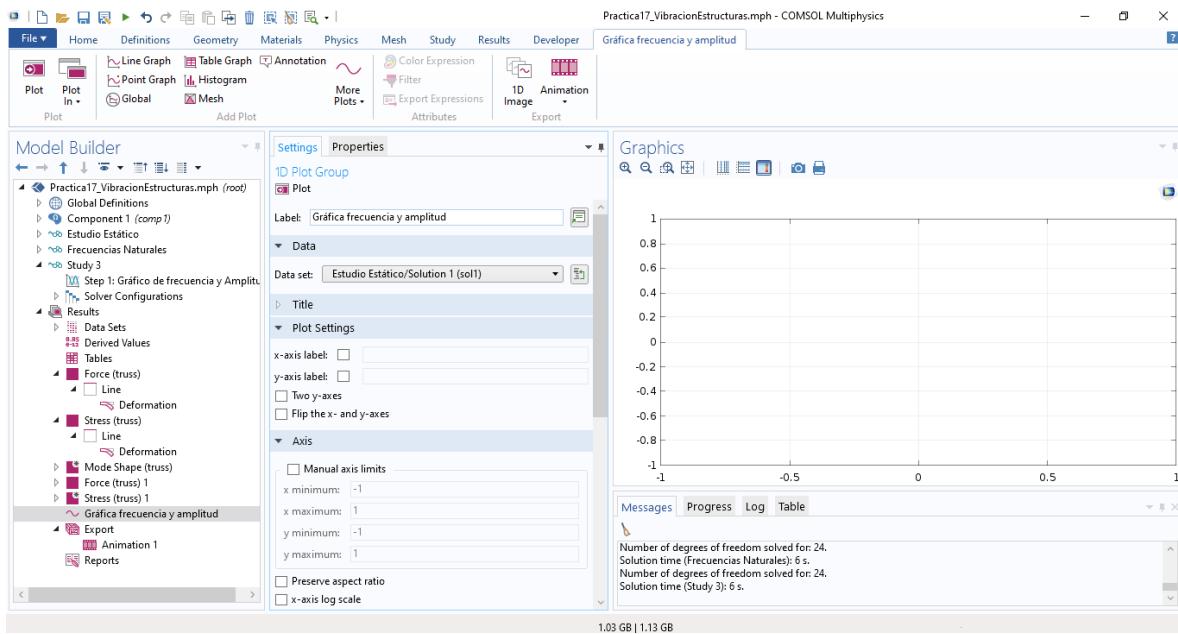




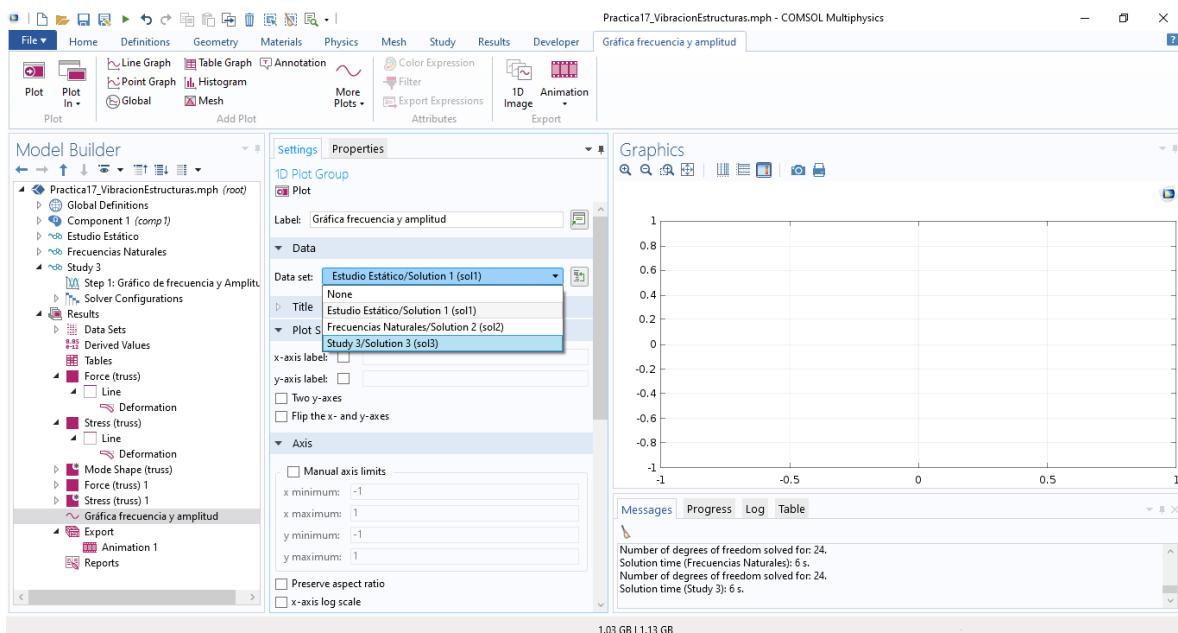
## GRÁFICA DE FRECUENCIAS ARMÓNICAS:

Para crear una gráfica con los armónicos de las vibraciones de los nodos hago lo siguiente:

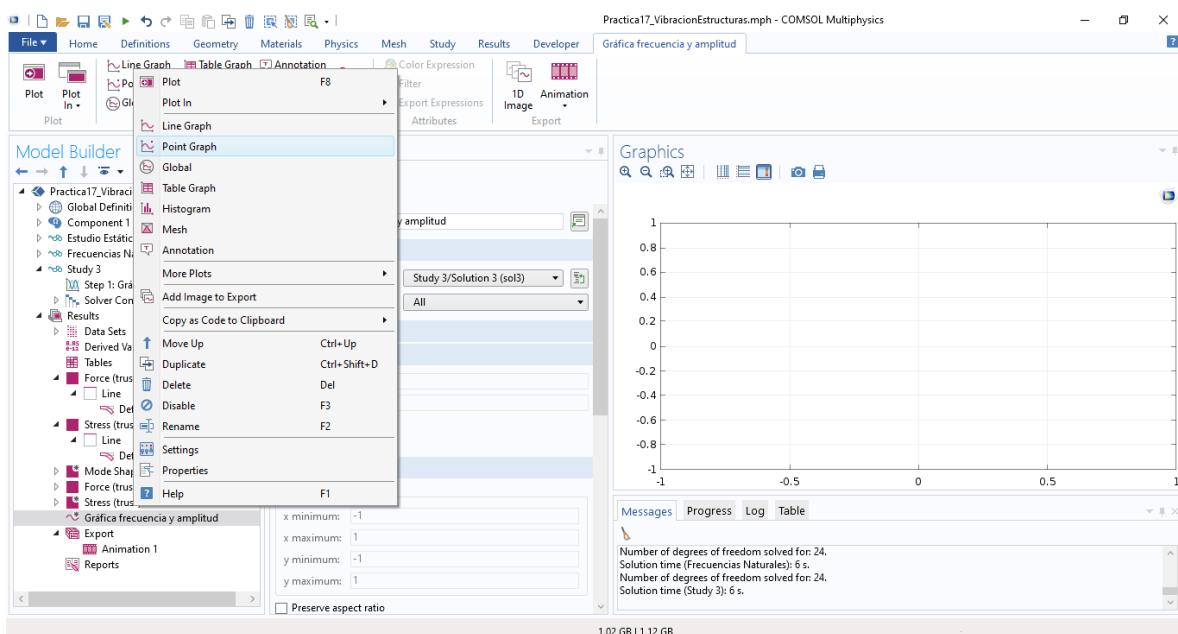
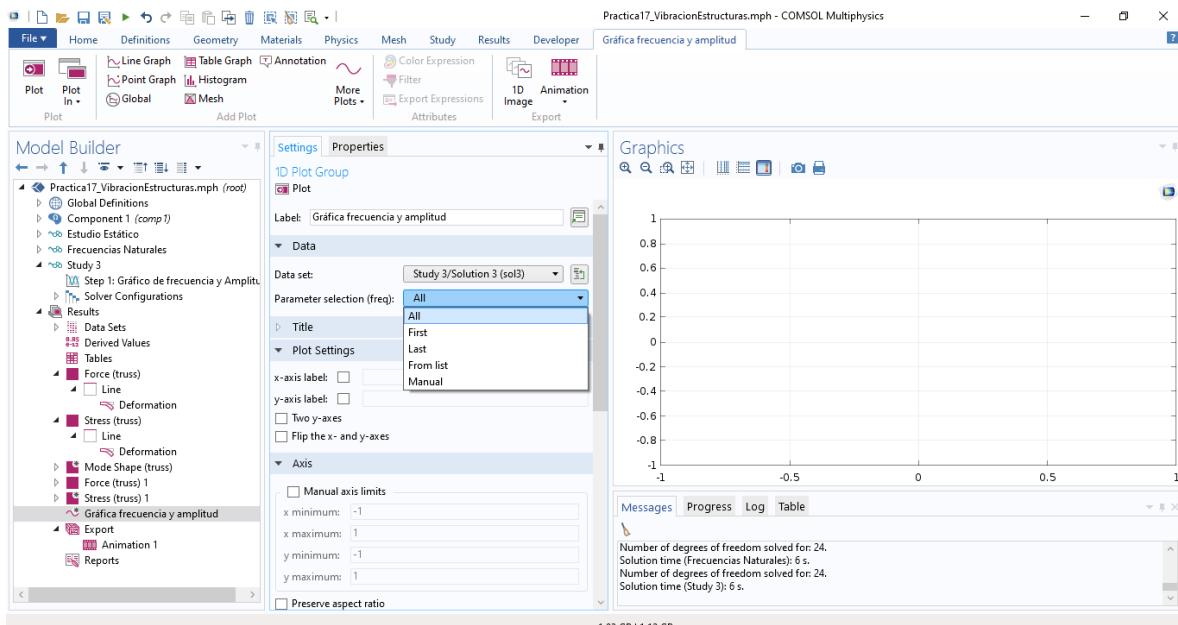




Luego selecciono el estudio que quiero graficar.

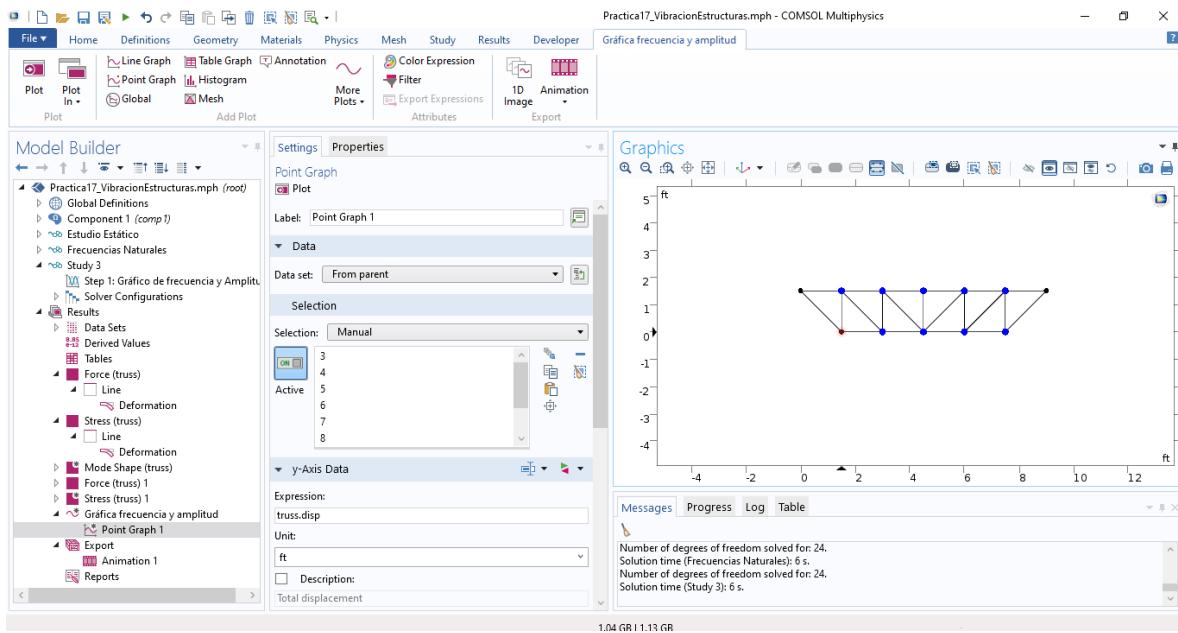
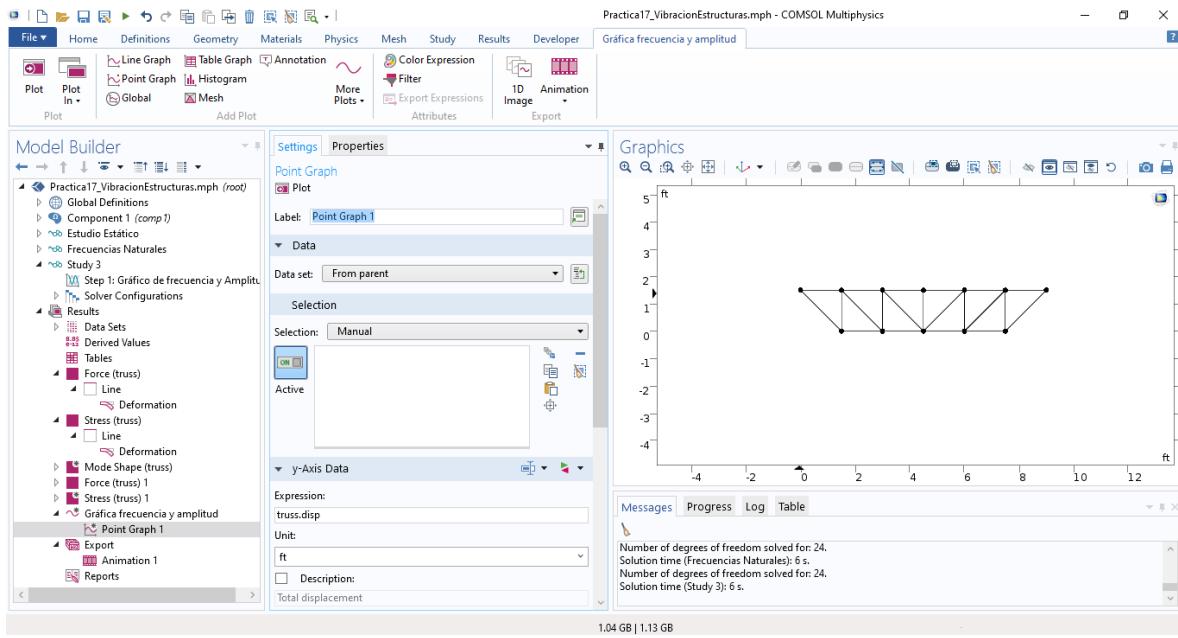


Después hay que indicar que resultados se van a poner en el gráfico.

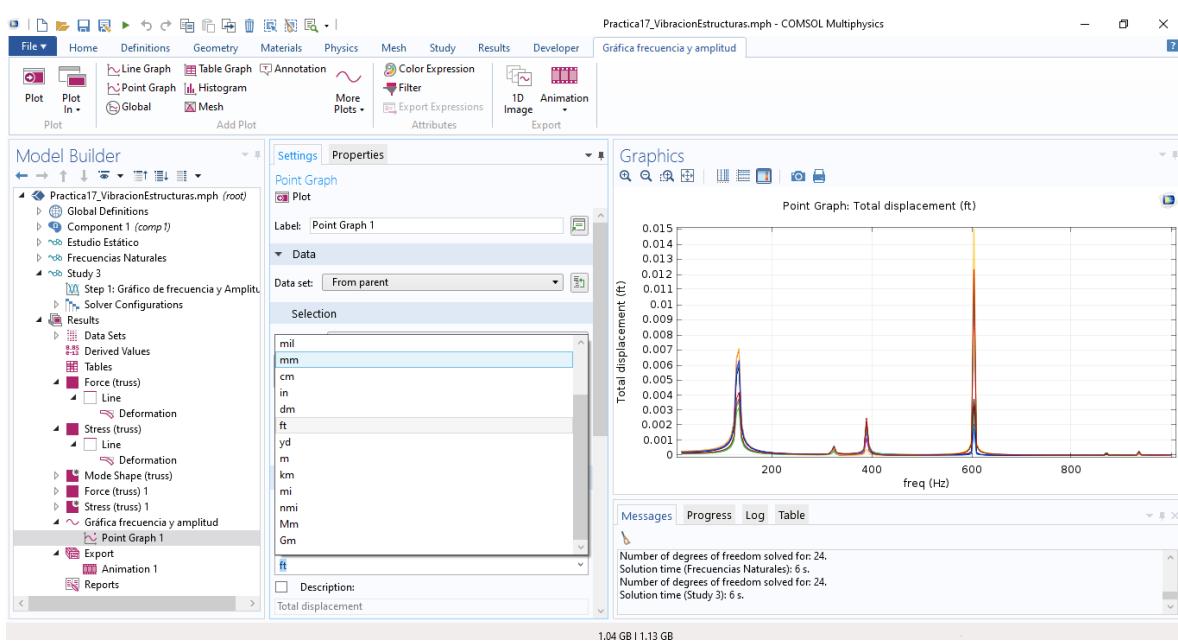
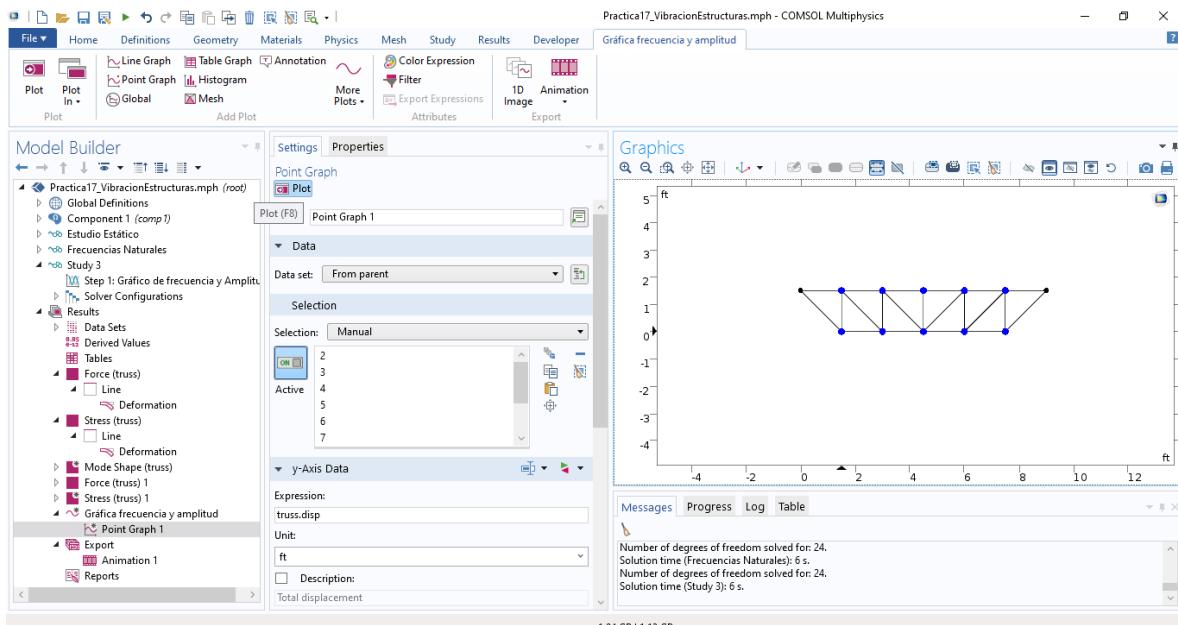


Luego de seleccionar en qué punto quiero que se grafique la frecuencia natural contra la amplitud del movimiento de la estructura.



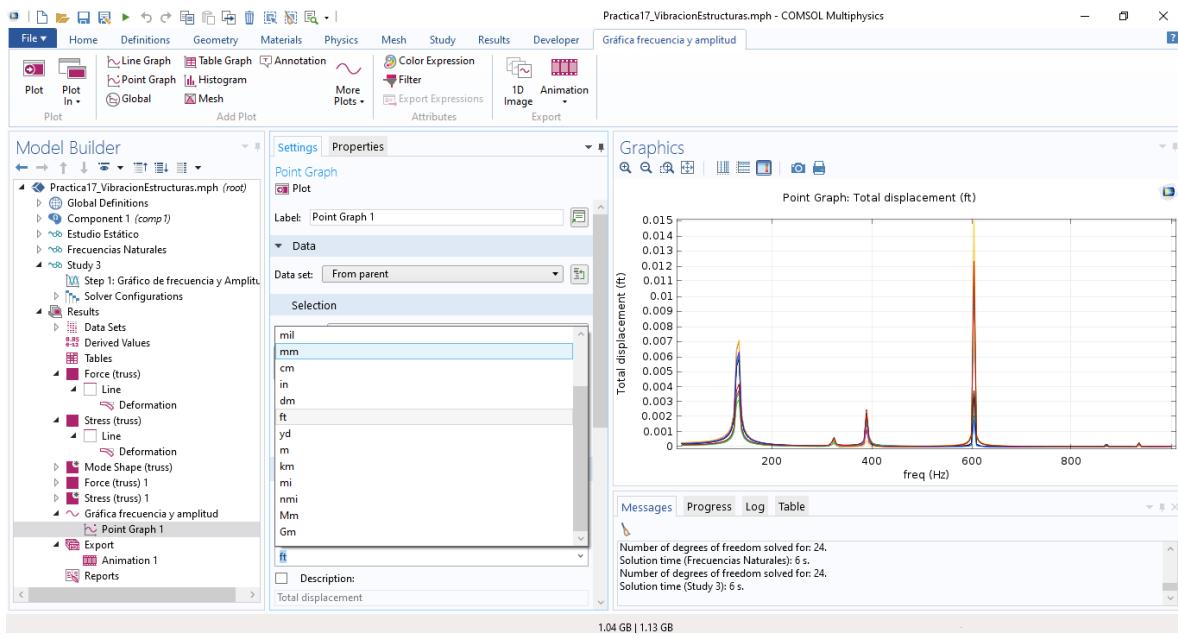


Puedo seleccionar uno, varios o todos los nodos.

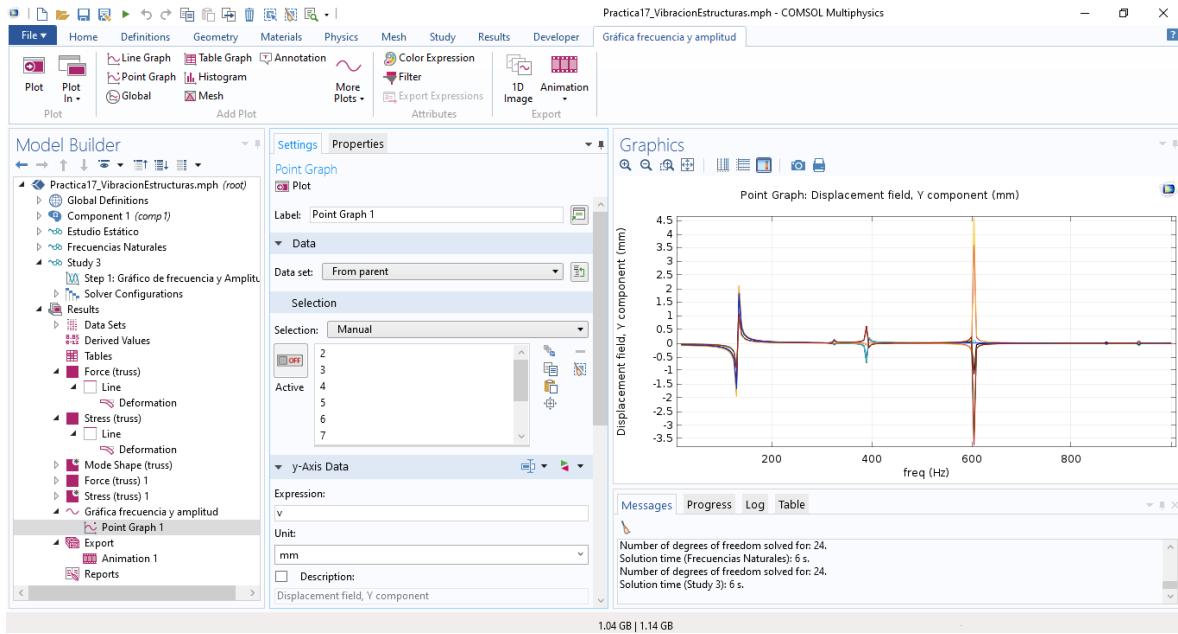


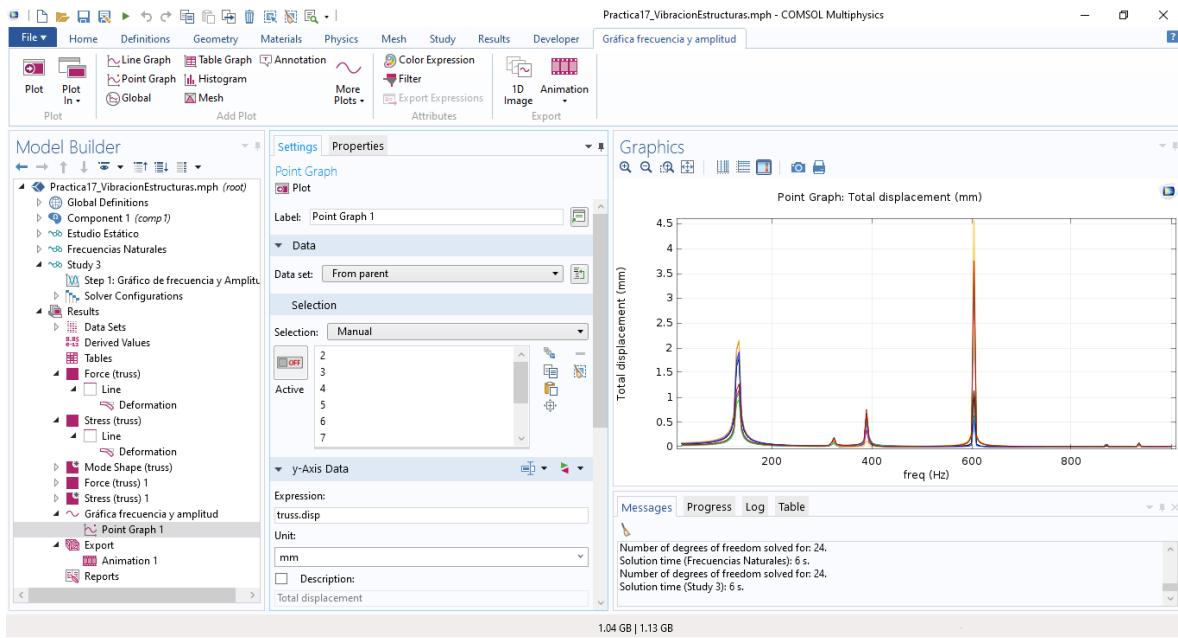
Es posible seleccionar la unidad del eje vertical que muestra el desplazamiento de los nodos en mi estructura.

En la gráfica de frecuencia los picos lo que representan son los armónicos.



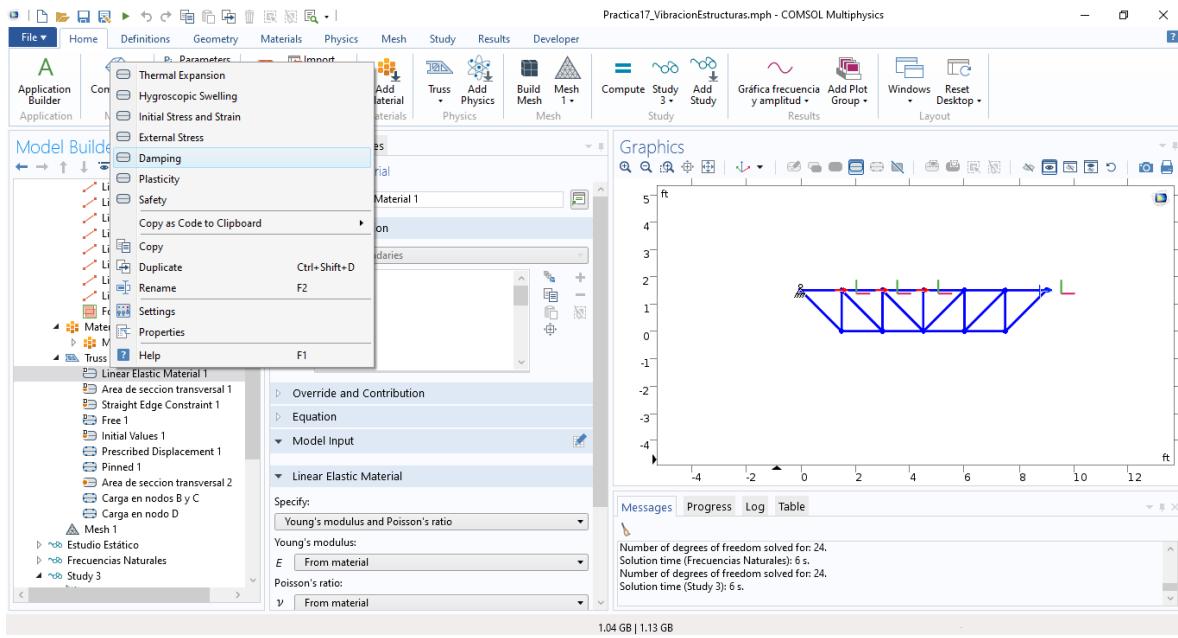
Con la instrucción ***truss.disp*** lo que se grafica es el desplazamiento total, pero puedo poner igual v o u para que se vea el desplazamiento vertical u horizontal, esto se verá en forma asintótica porque el desplazamiento puede ser positivo y negativo hacia el infinito.



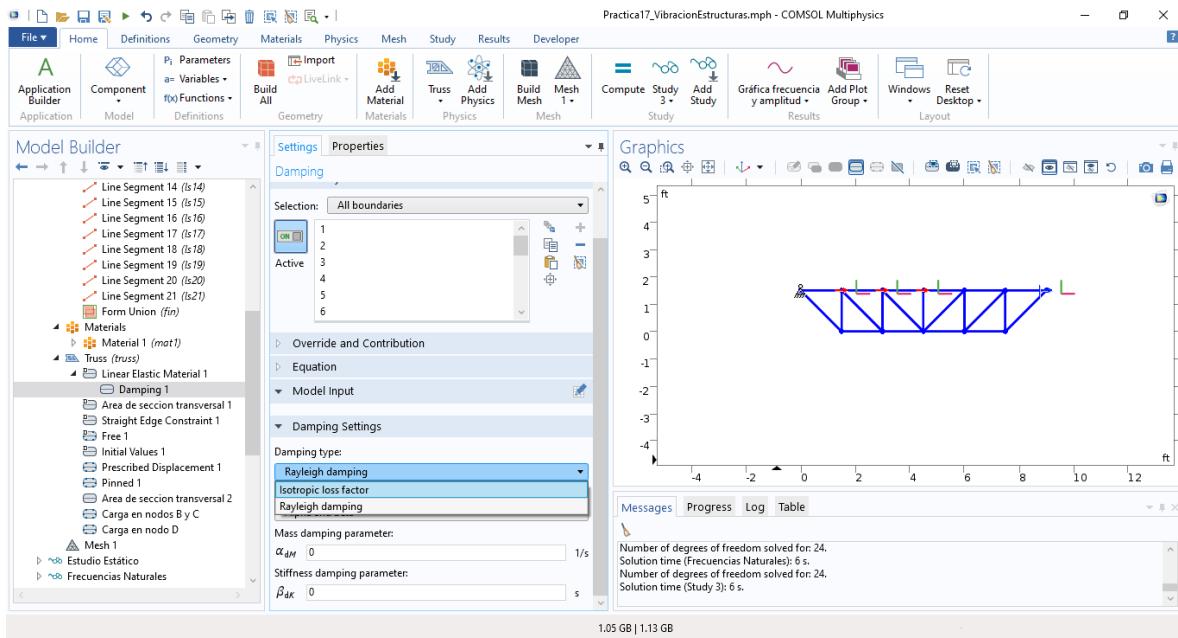


En la vida real esto no pasa tan exagerado porque el material tiene una propiedad de disipación de energía que funciona como un tipo amortiguador.

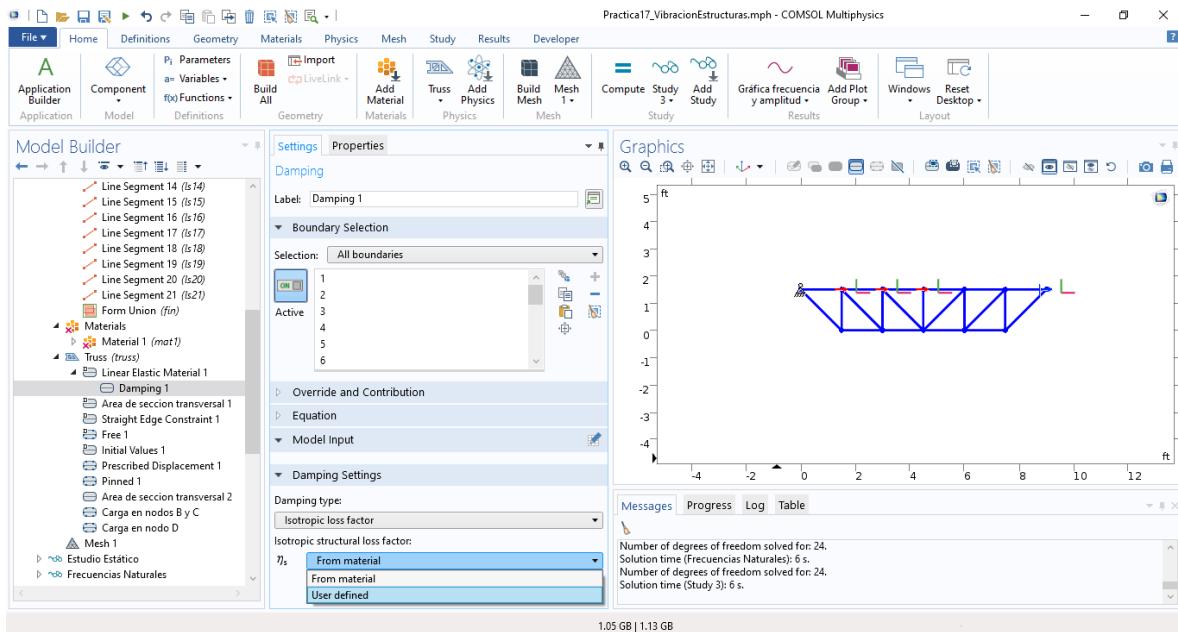
Esto se agrega en la estructura dentro de Truss con la instrucción Damping.



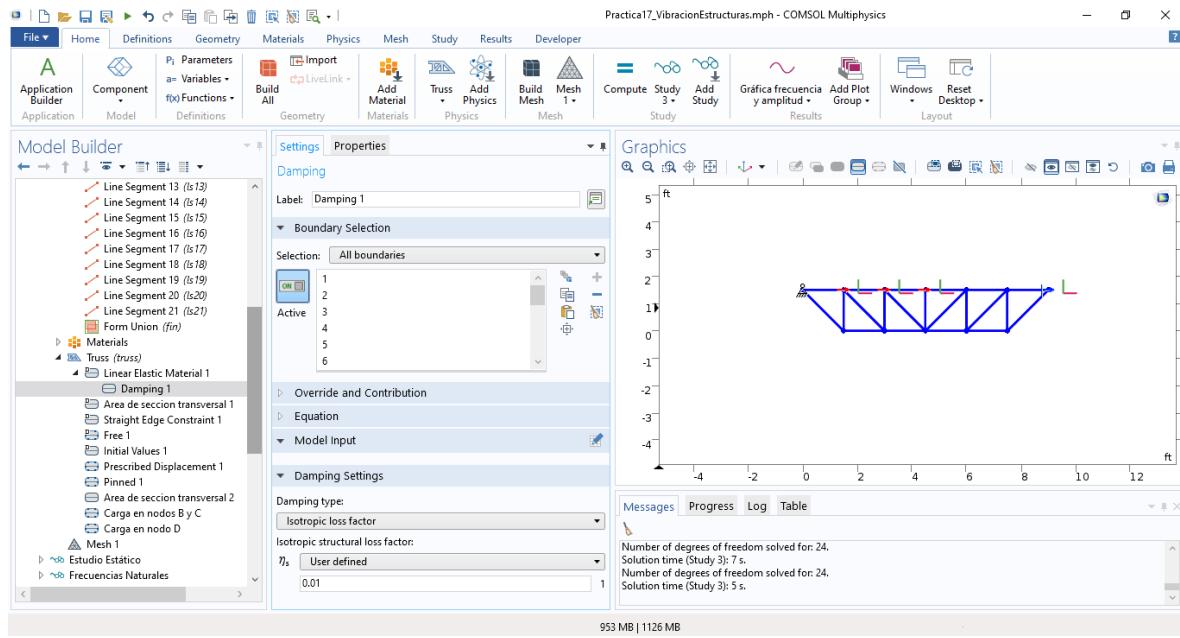
El amortiguamiento del material en el programa se mete como Isotropic loss factor.



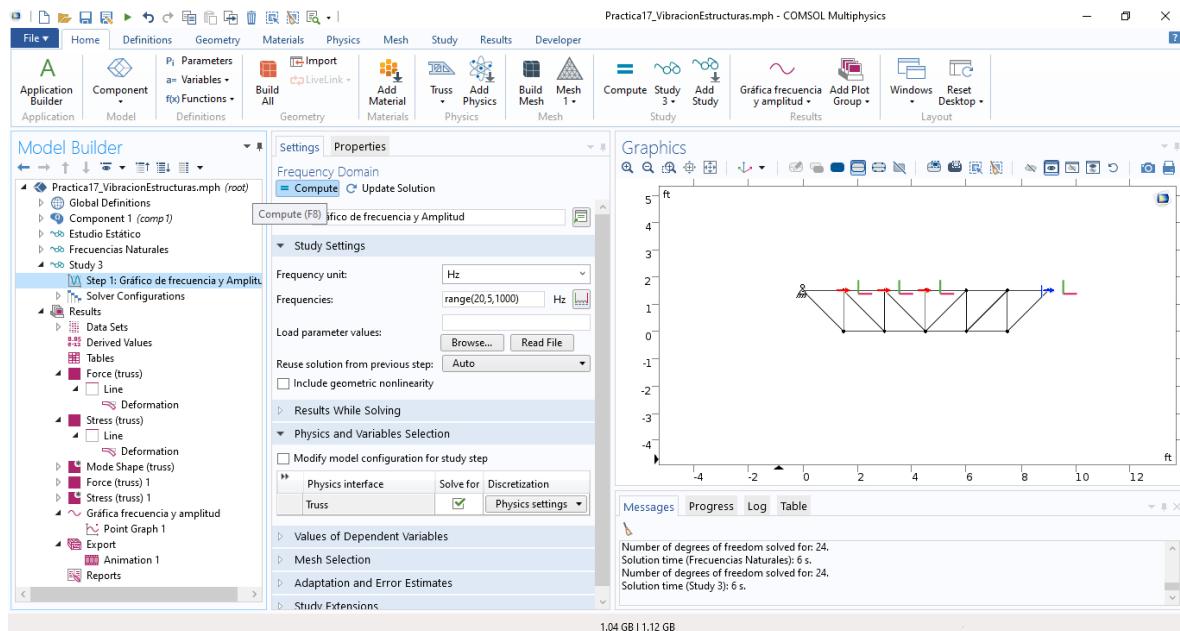
Este es isotrópico y lo que significa es que sus propiedades no cambiarán no importando el punto en el que lo analice, esto en la vida real pasa con el acero.

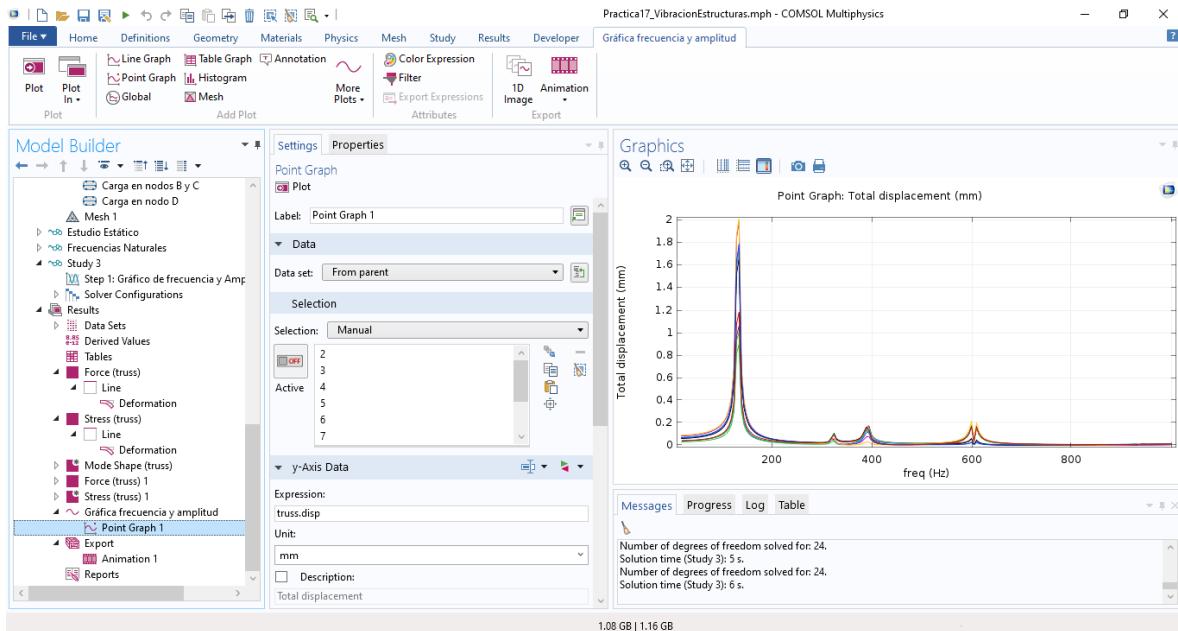
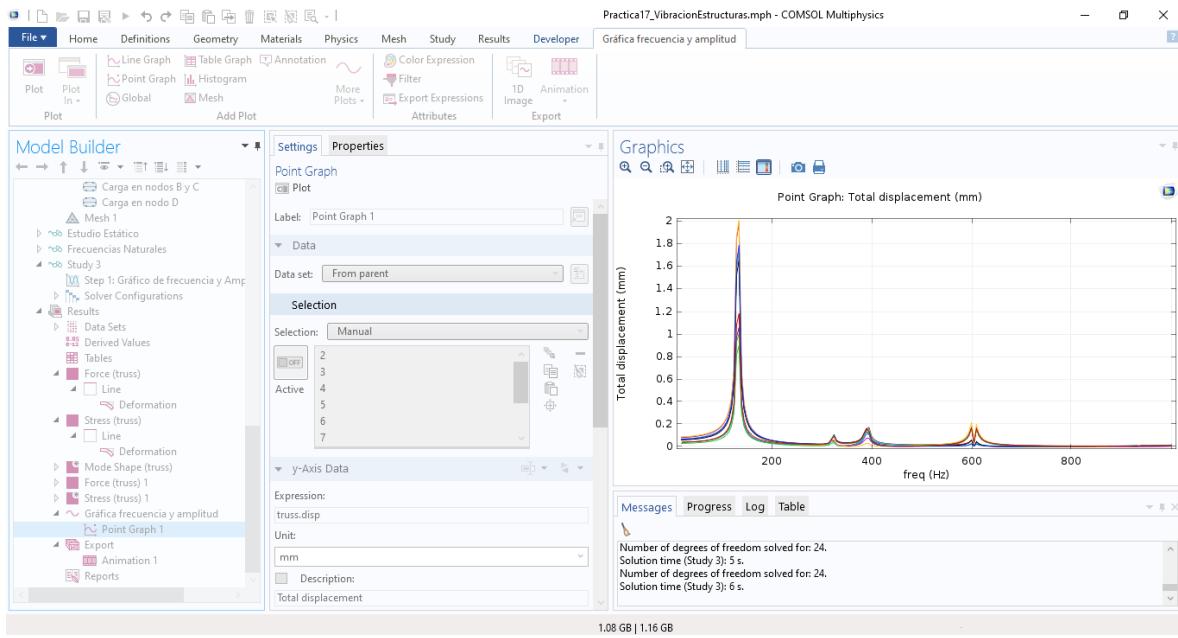


En el acero la amortiguación es de 0.01.

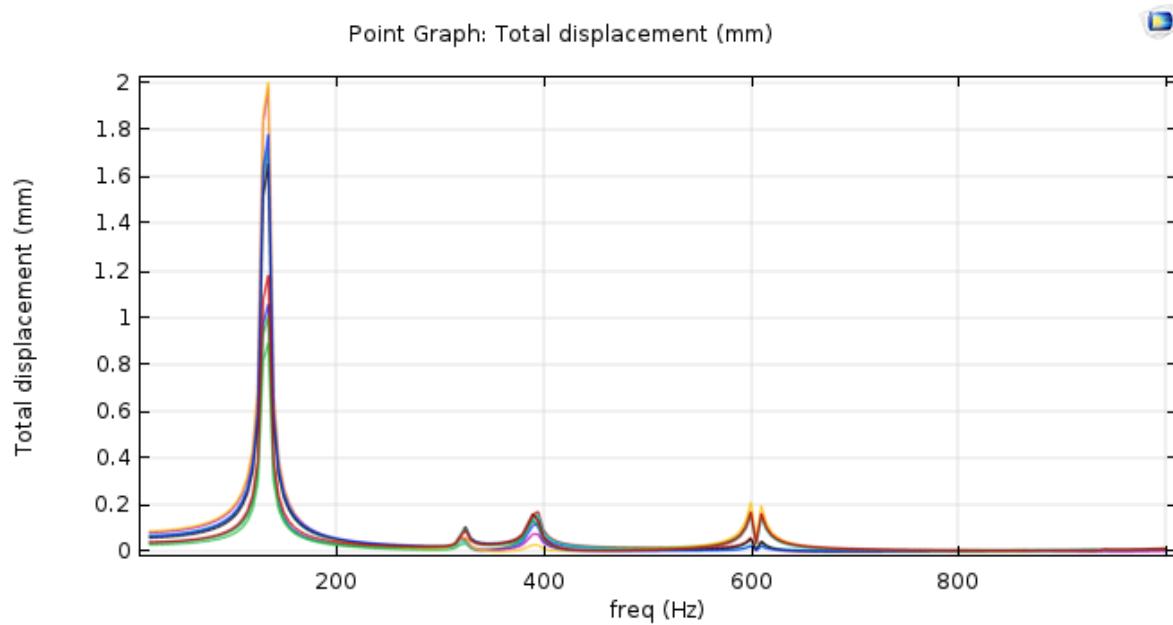


Y vuelvo a hacer el cálculo de el gráfico de frecuencia.





Si quiero guardar la imagen le debo dar clic a la camarita de arriba de la gráfica y solo pegarla en el documento con CRTL+V.



## BIBLIOGRAFÍA:

INGENIERÍA MECÁNICA ESTÁTICA (12VA EDICIÓN) – RUSSELL C. HIBBELER.