

INGENIERÍA MECATRÓNICA



DI_CERO

DIEGO CERVANTES RODRÍGUEZ

INGENIERÍA ASISTIDA POR COMPUTADORA

COMSOL MULTIPHYSICS 5.6

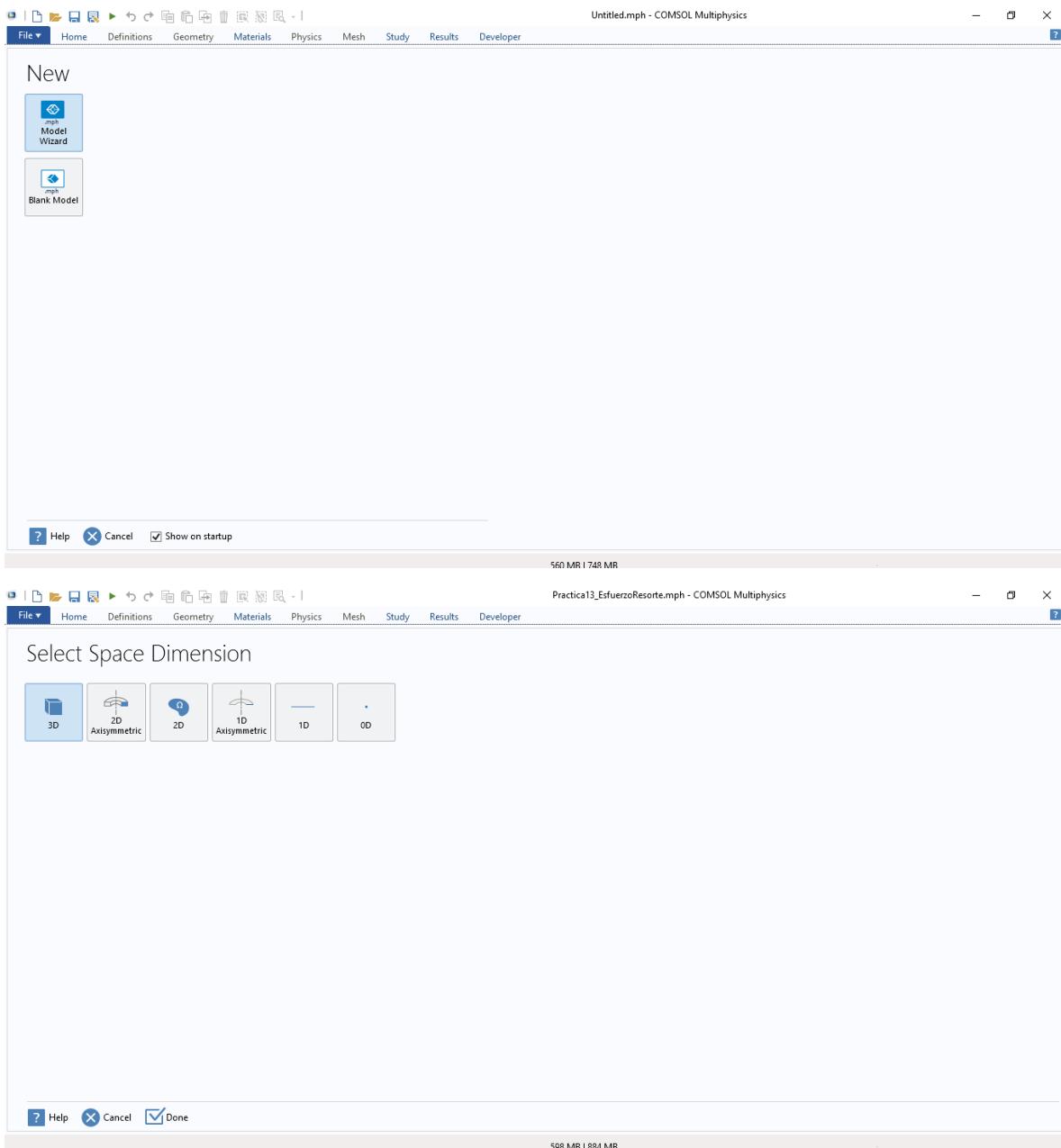
14: Esfuerzo de Compresión
en un Resorte

Contenido

CREACIÓN DE LA PIEZA EN COMSOL:.....	2
ANÁLISIS MECÁNICO EN COMSOL:	12
RESULTADO DEL ELEMENTO FINITO EN COMSOL:	18
BIBLIOGRAFÍA:.....	27

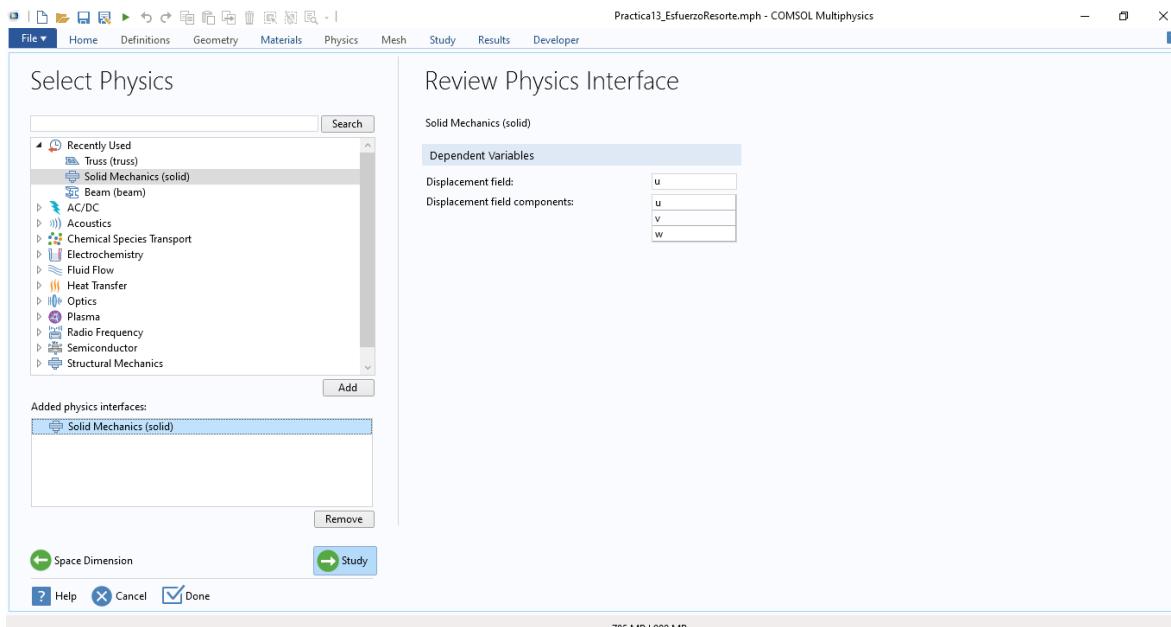
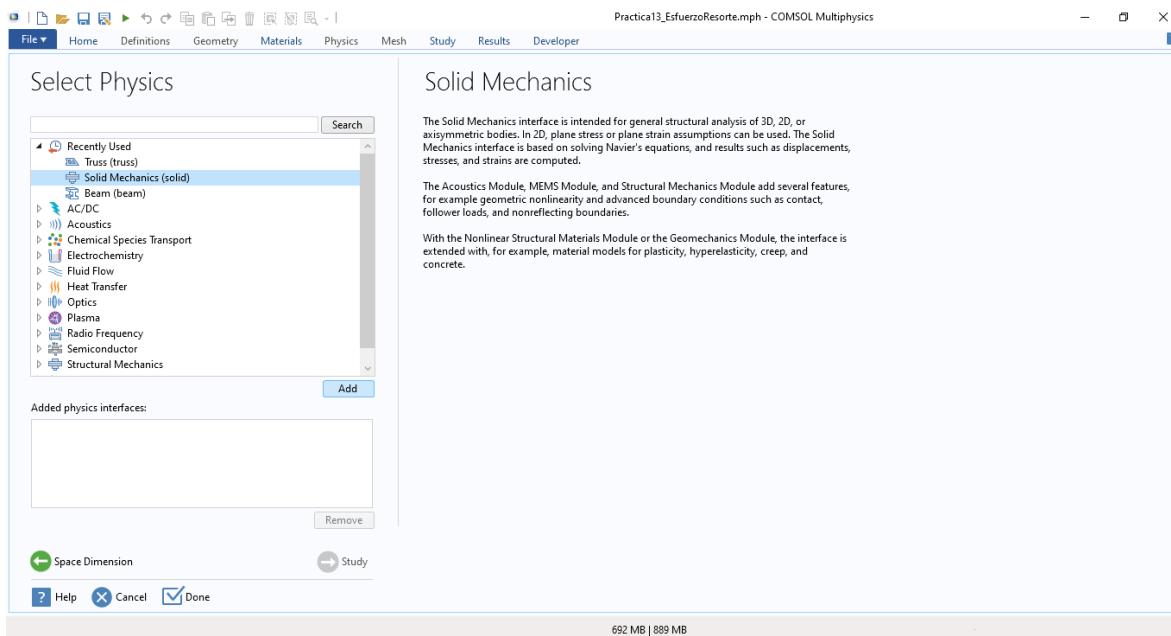


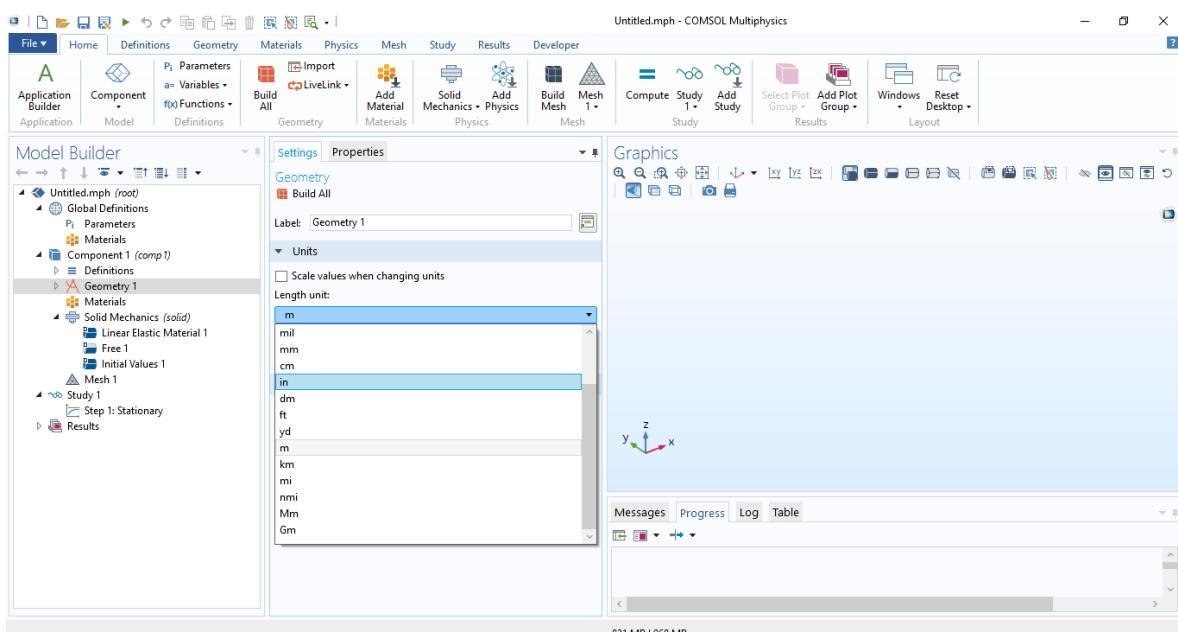
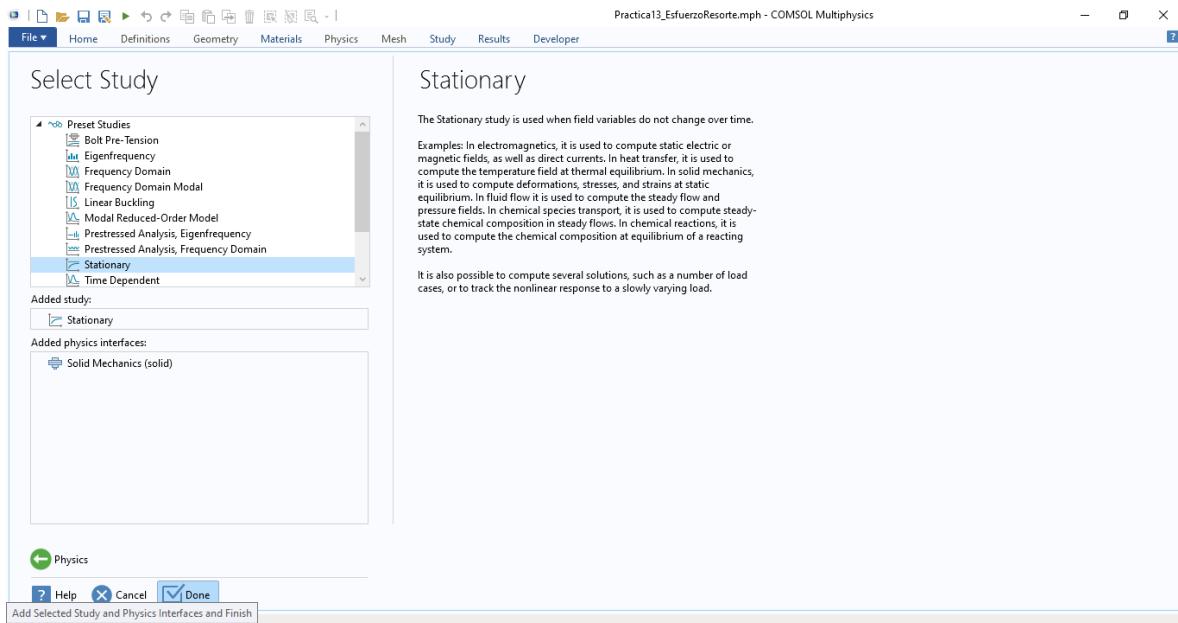
CREACIÓN DE LA PIEZA EN COMSOL:

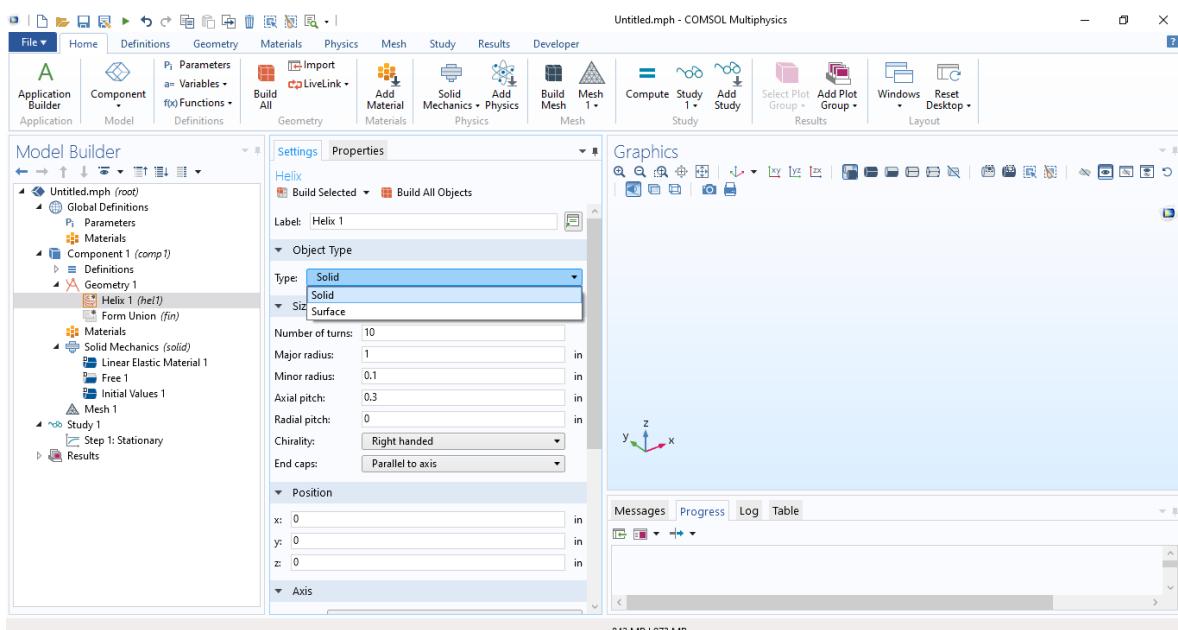
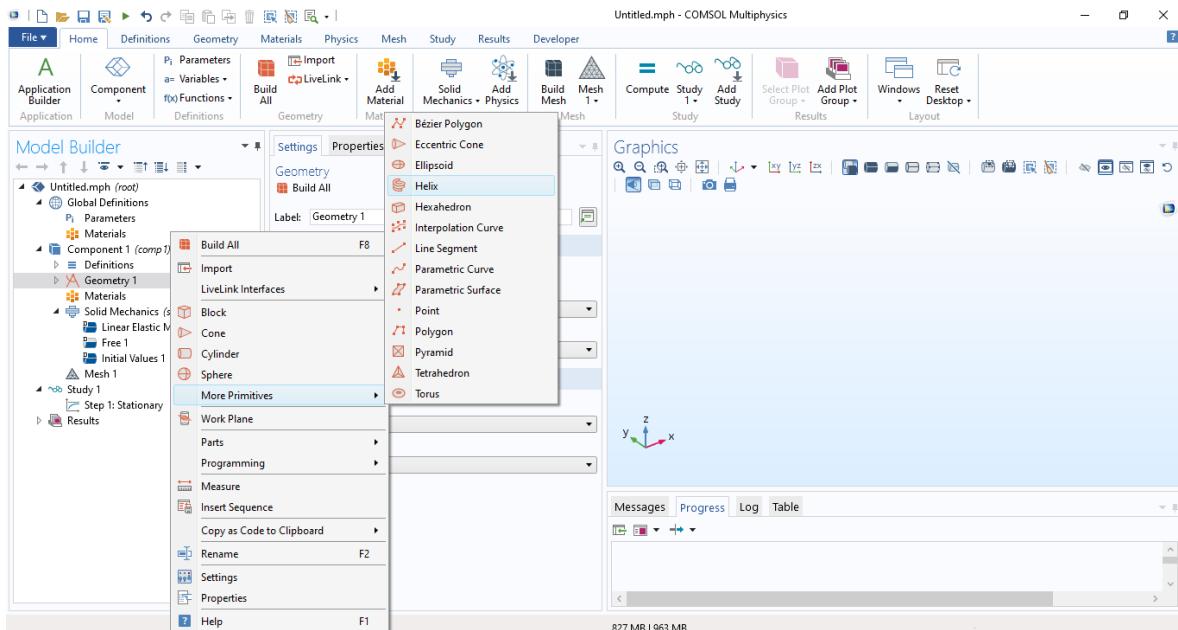


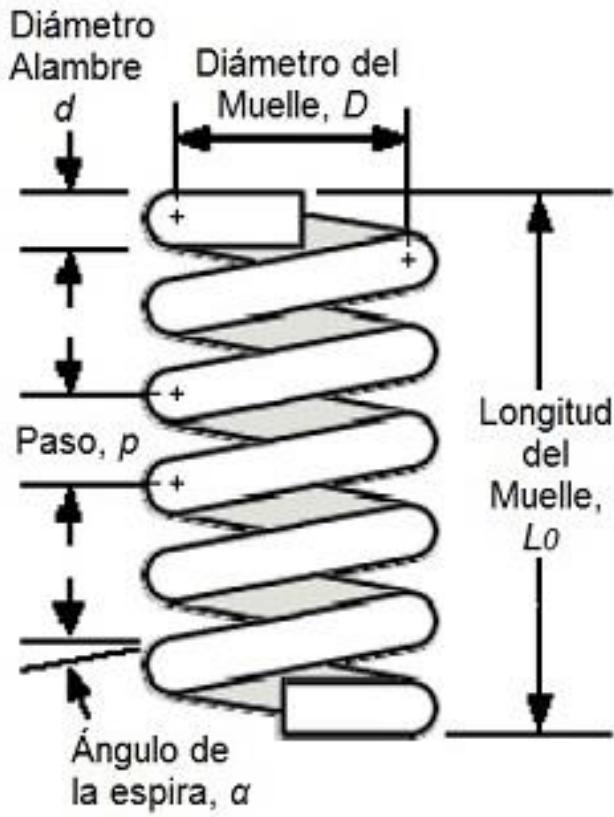
2

DI_CERO









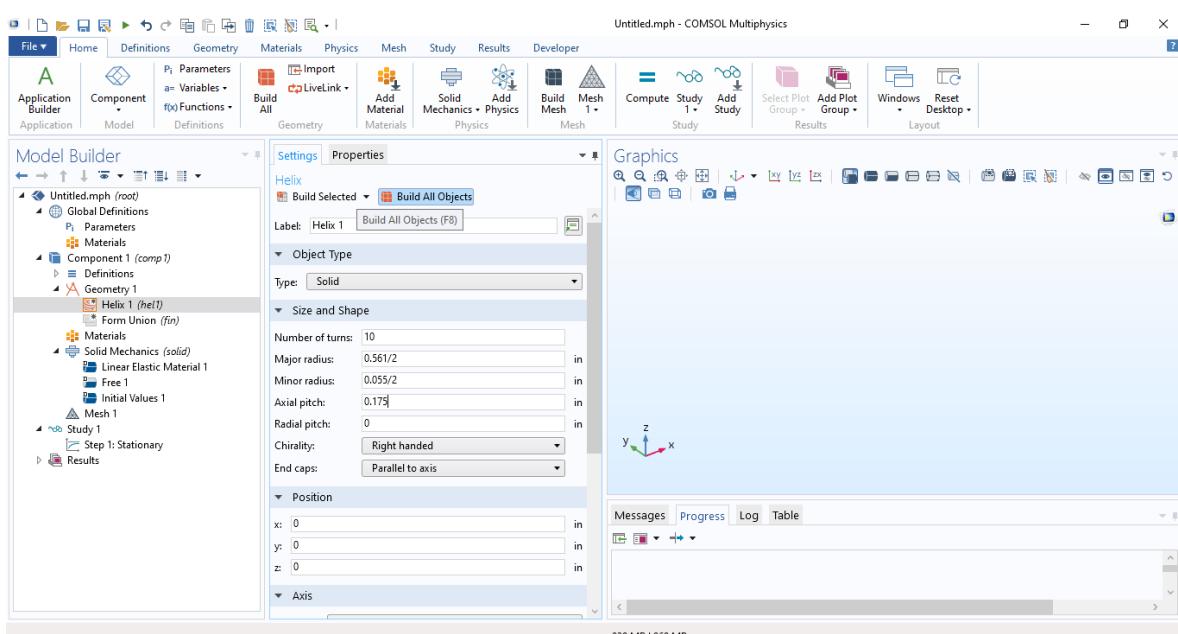
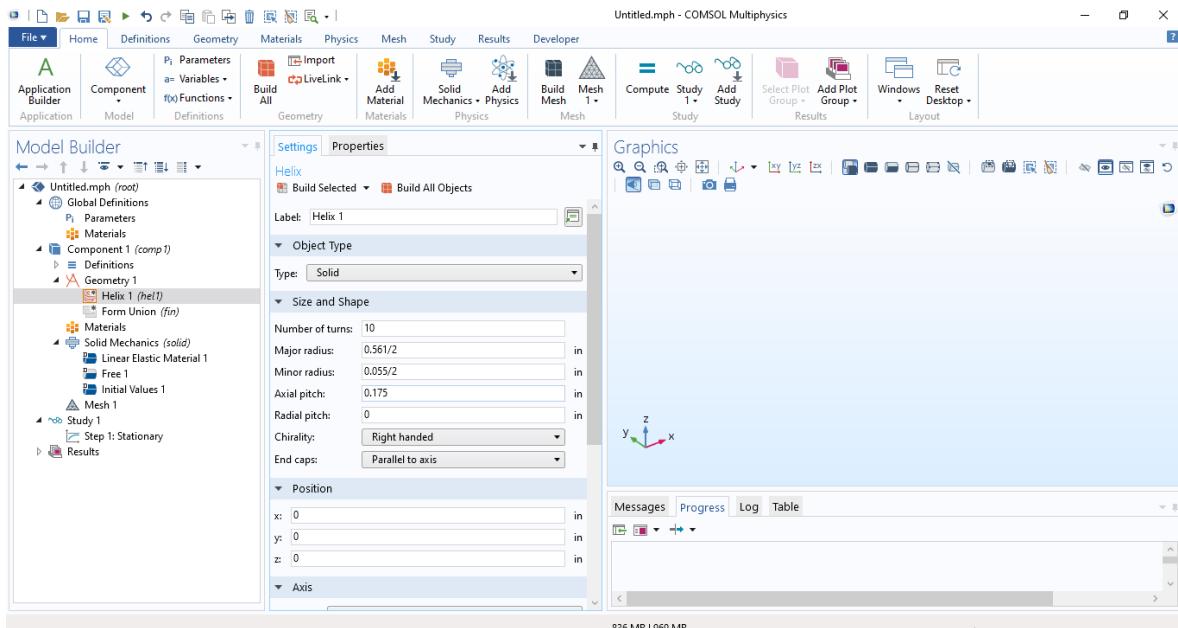
La fórmula para obtener el *Major radius* del resorte es: $\frac{Dr}{2} = \frac{D}{2}$

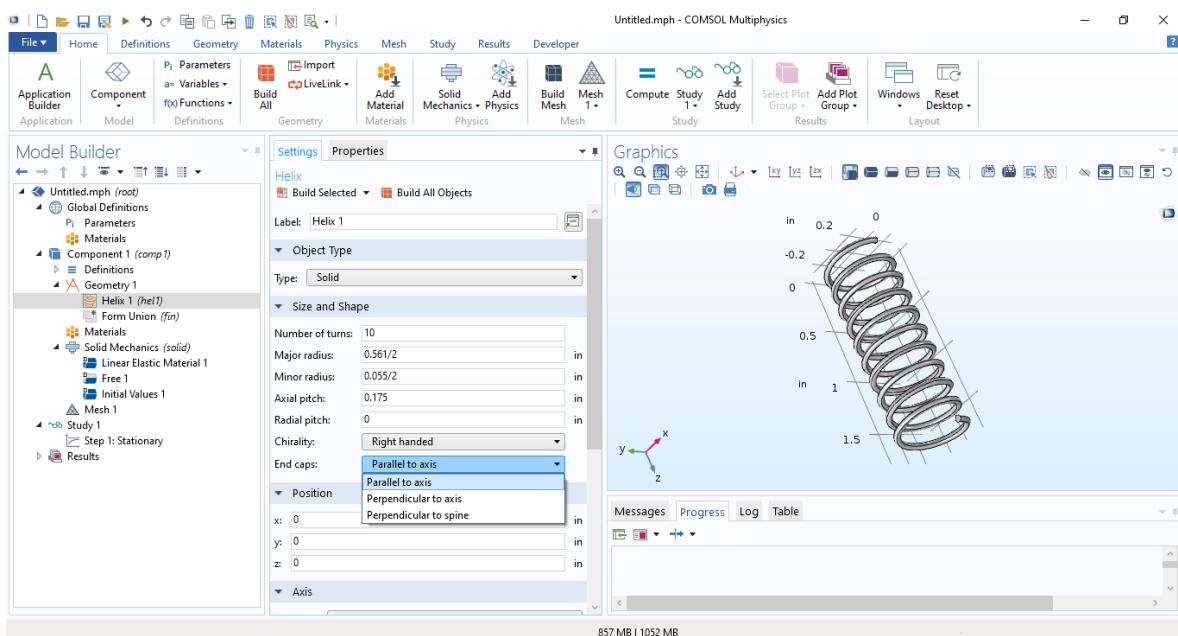
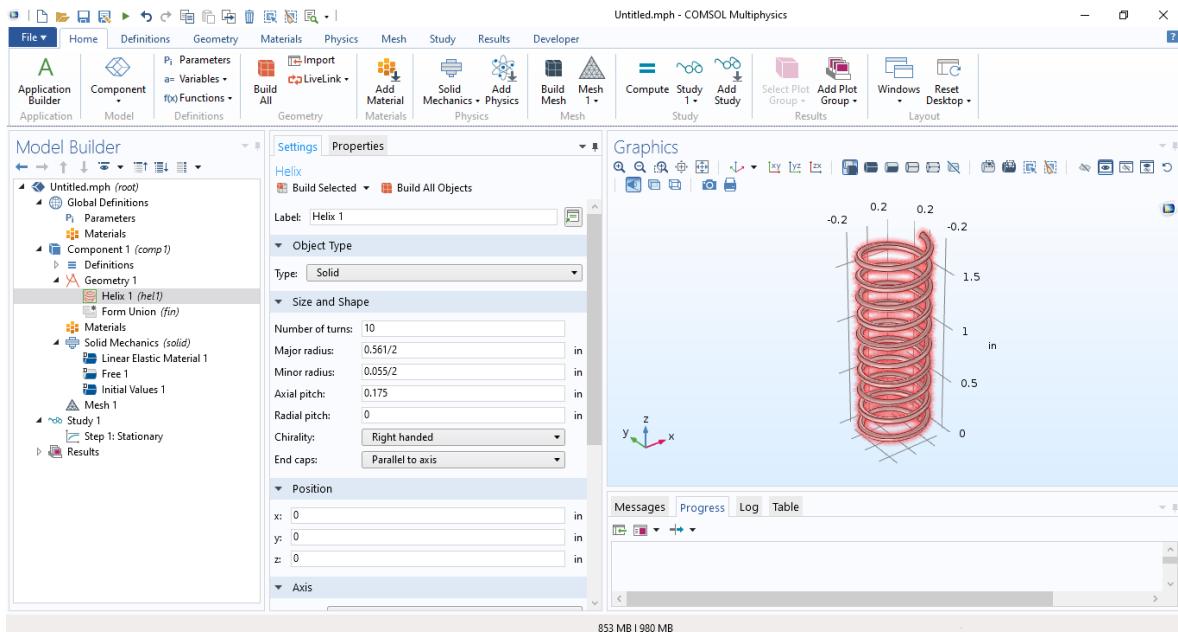
La fórmula para obtener el *Minor radius* del resorte es: $\frac{Dw}{2} = \frac{d}{2}$

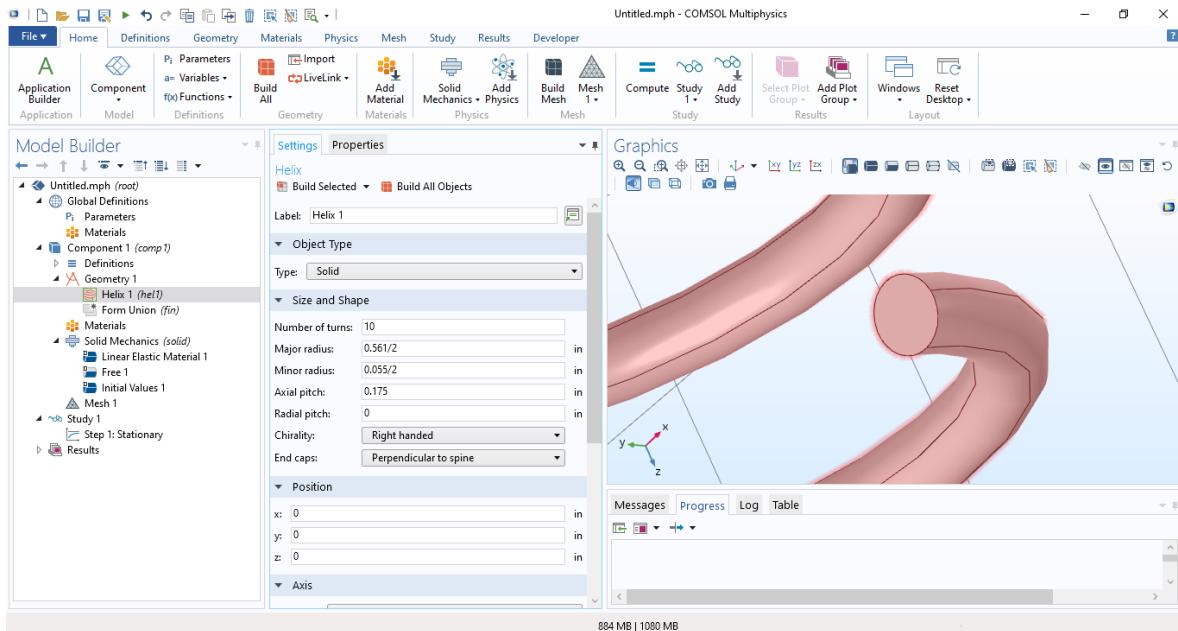
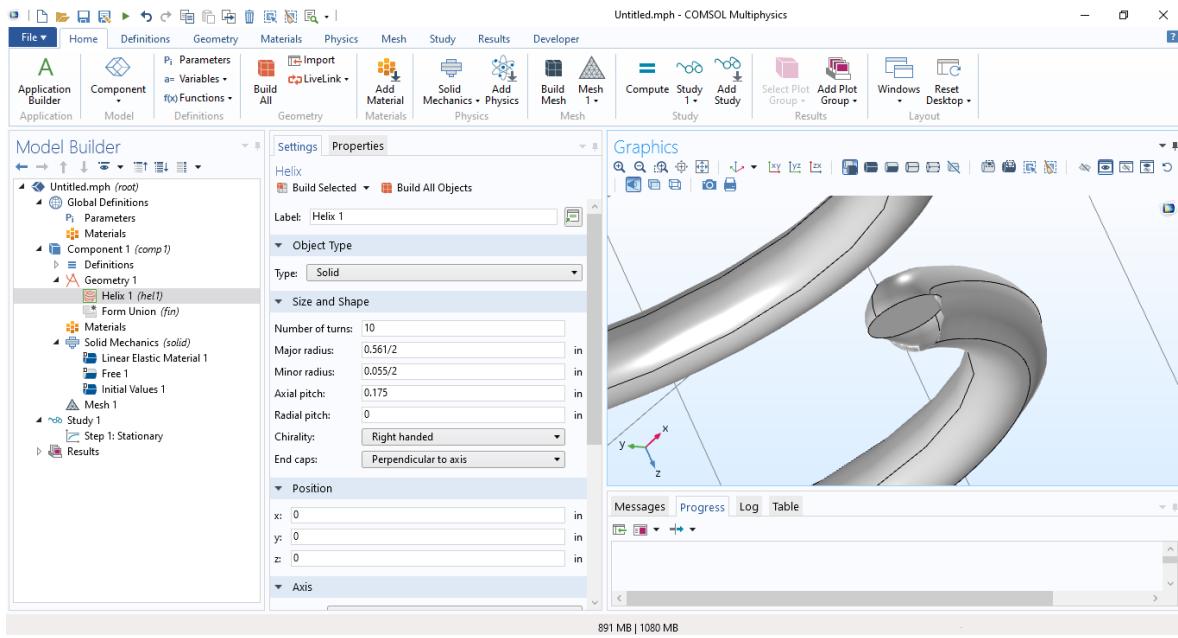
The screenshot shows the COMSOL Multiphysics interface with the following details:

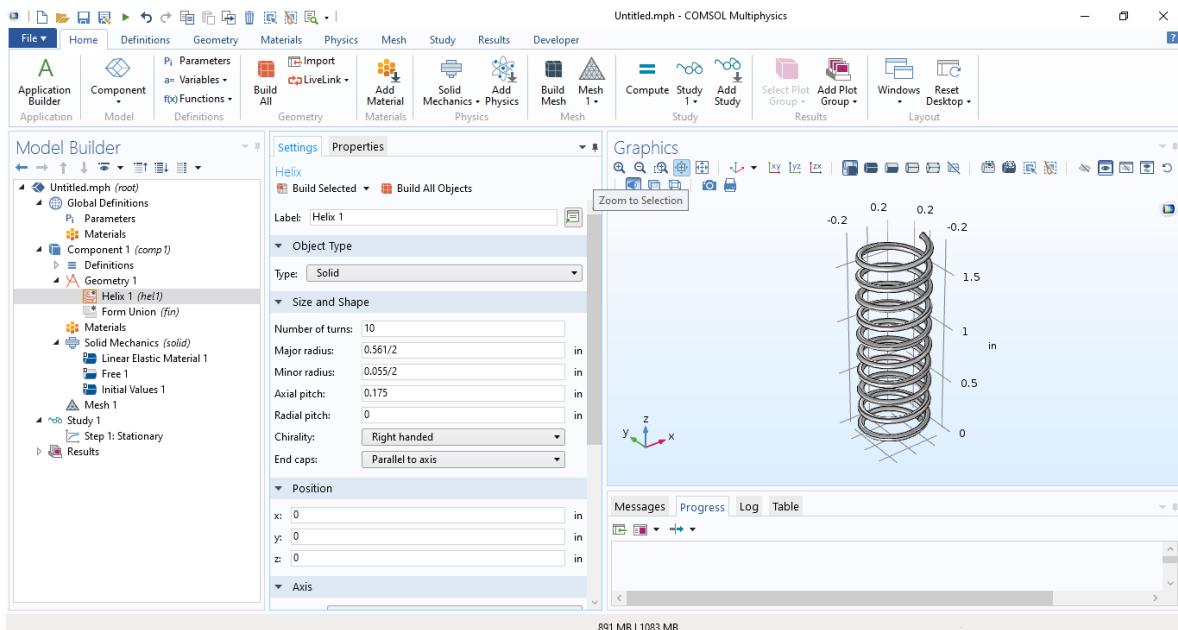
- Model Builder** (left panel):
 - Project tree: Untitled.mph (root) > Global Definitions > Parameters, Materials.
 - Component 1 (comp1) > Geometry 1 > Helix 1 (selected).
 - Properties for Helix 1:
 - Label: Helix 1
 - Type: Solid
 - Number of turns: 10
 - Major radius: 0.561/2 in
 - Minor radius: 0.055/2 in
 - Axial pitch: 0.3 in
 - Radial pitch: 0 in
 - Chirality: Right handed
 - End caps: Parallel to axis
 - Position settings: x: 0, y: 0, z: 0
 - Axis settings
- Graphics** (right panel):
 - 3D visualization of the helix part.
 - Coordinate system axes (x, y, z).
- Toolbar** (top):
 - File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, Developer.
 - Import, LiveLink, Add Material, Add Mechanics, Build Mesh, Compute, Select Plot, Windows, Reset Desktop.
- Status Bar** (bottom):
 - 836 MB | 969 MB

El paso P es: *Axial pitch*.

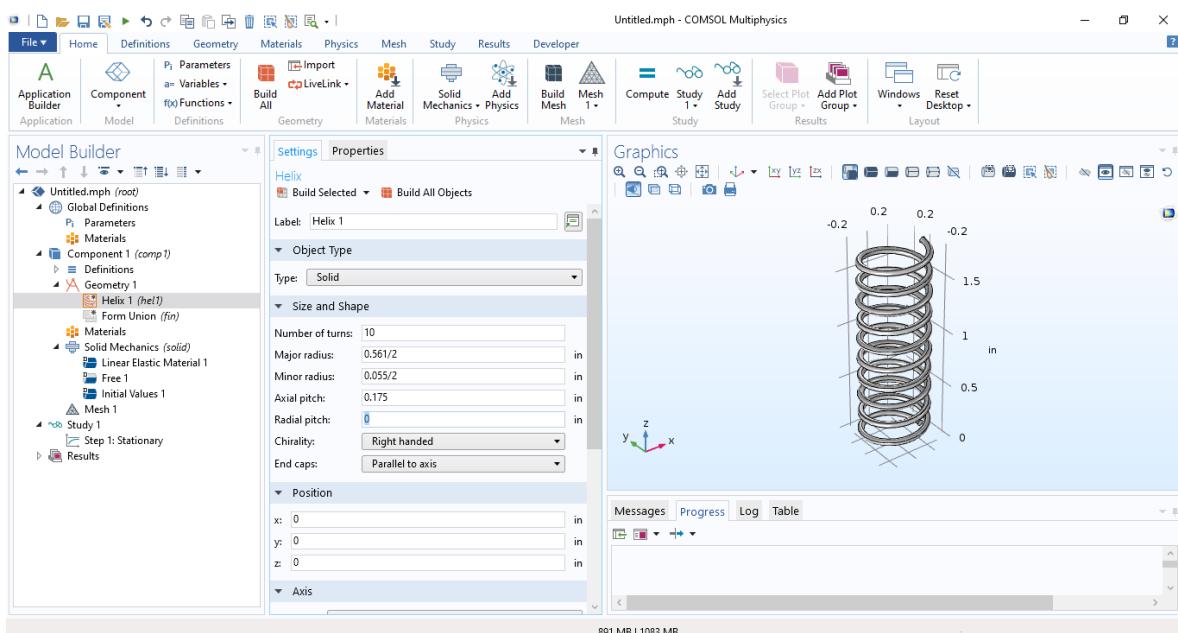


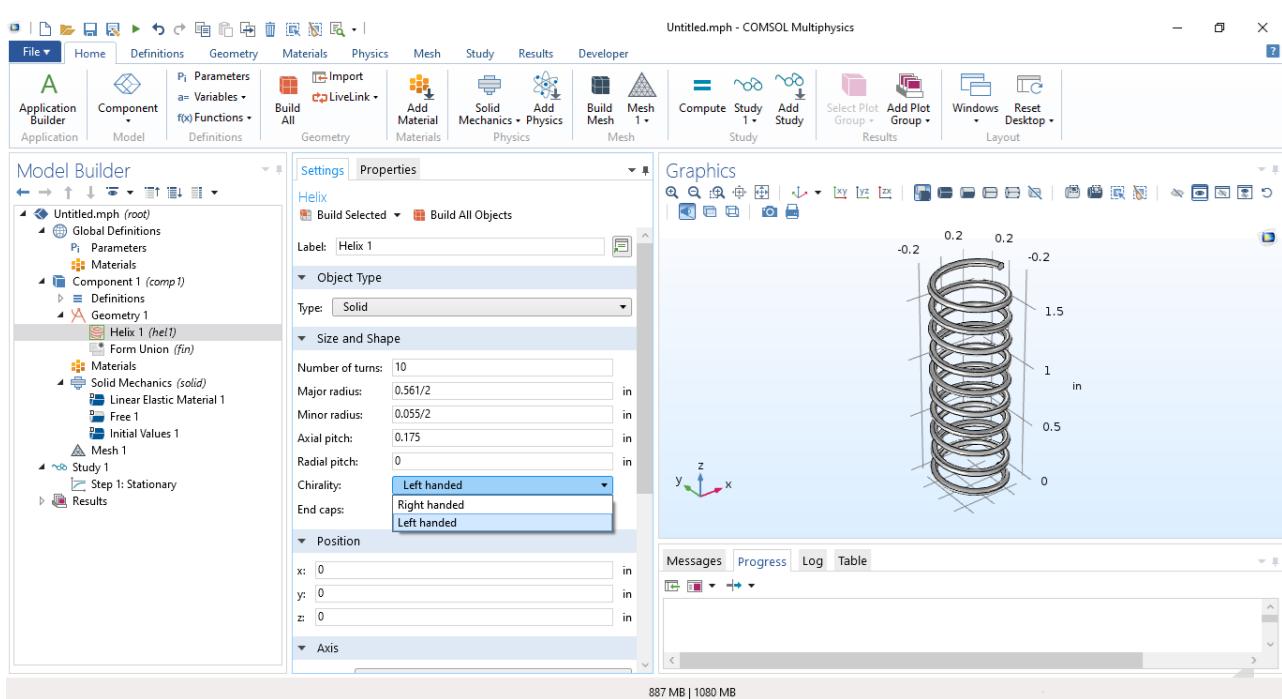
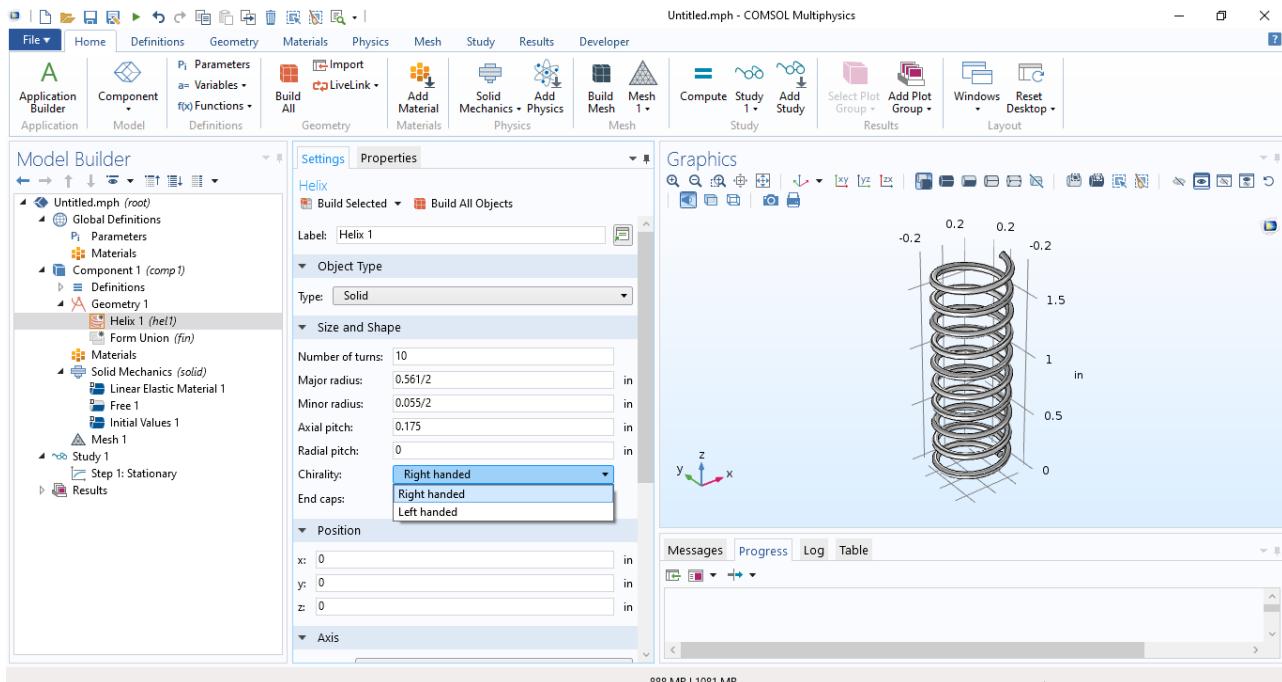






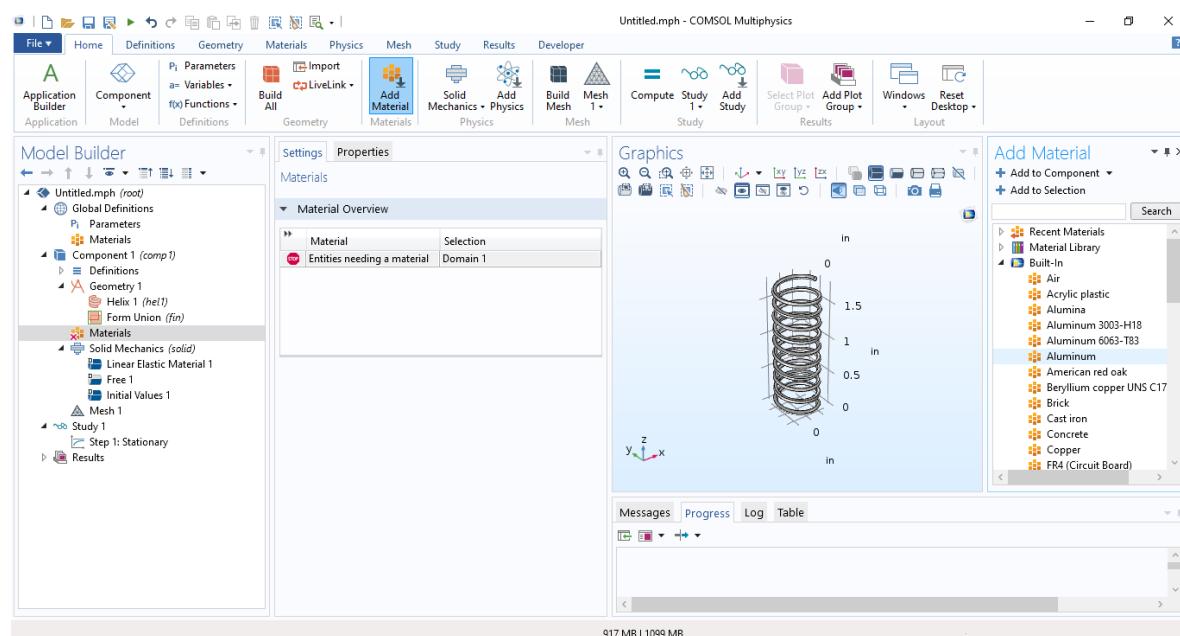
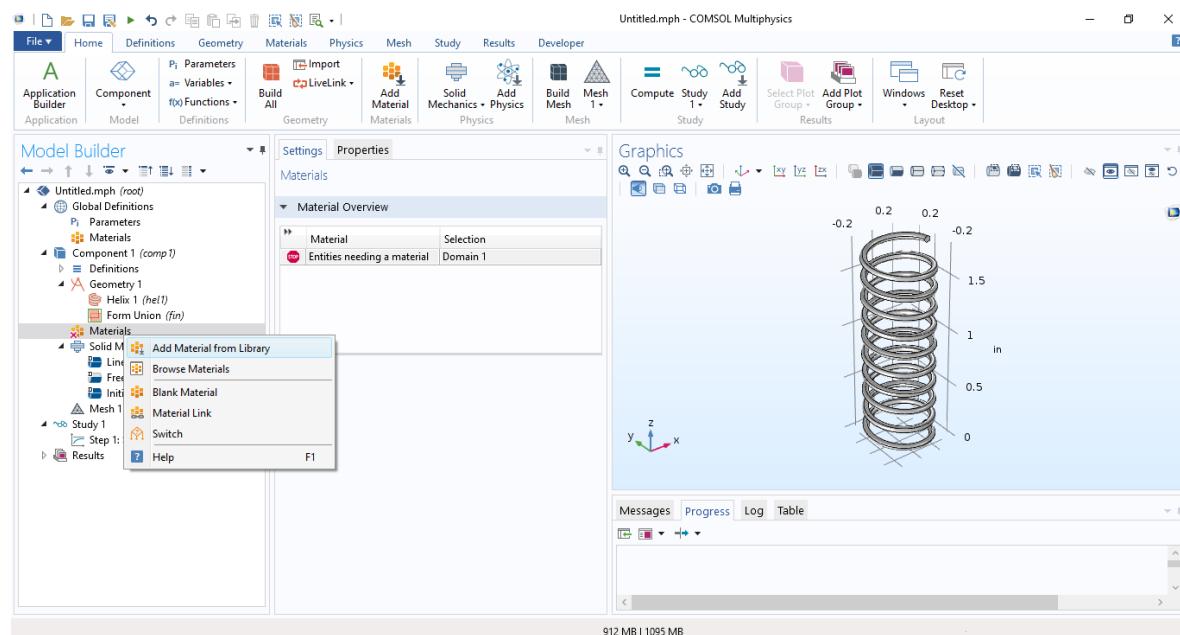
Si pongo algún valor en la variable de *Radial pitch*: hará que mi resorte tenga un radio variable.

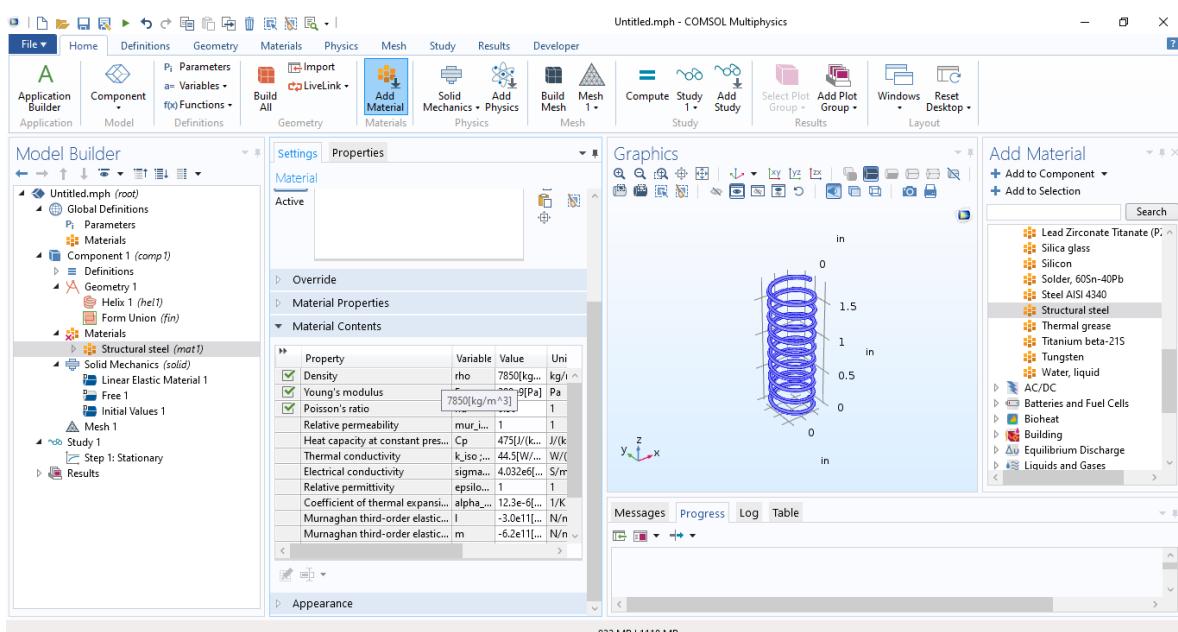
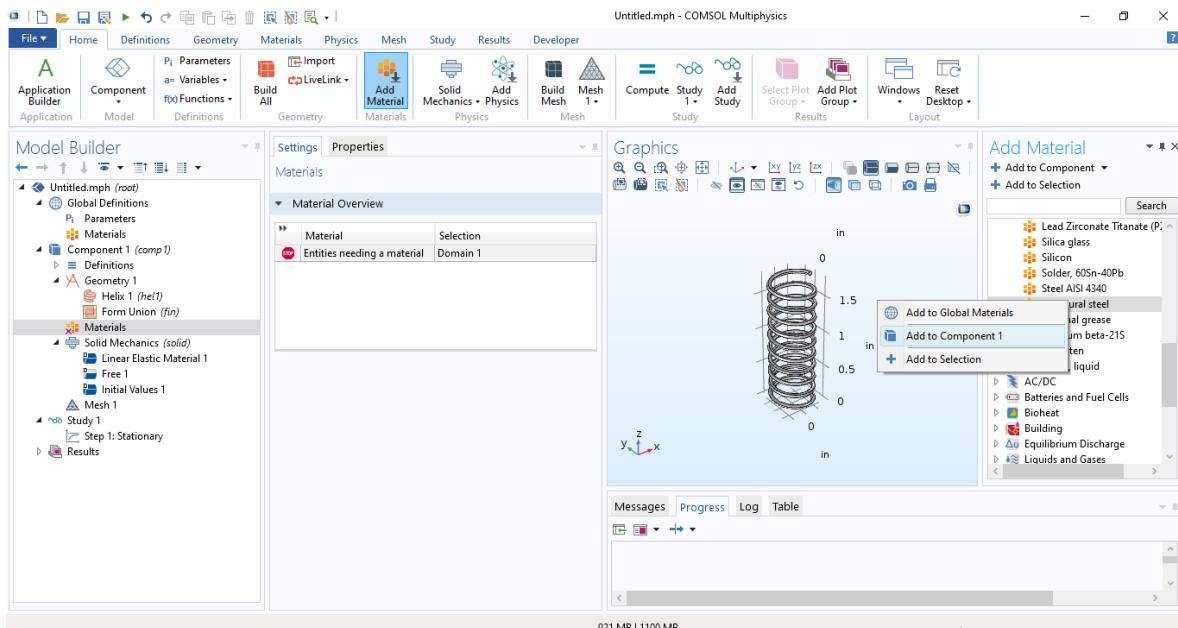


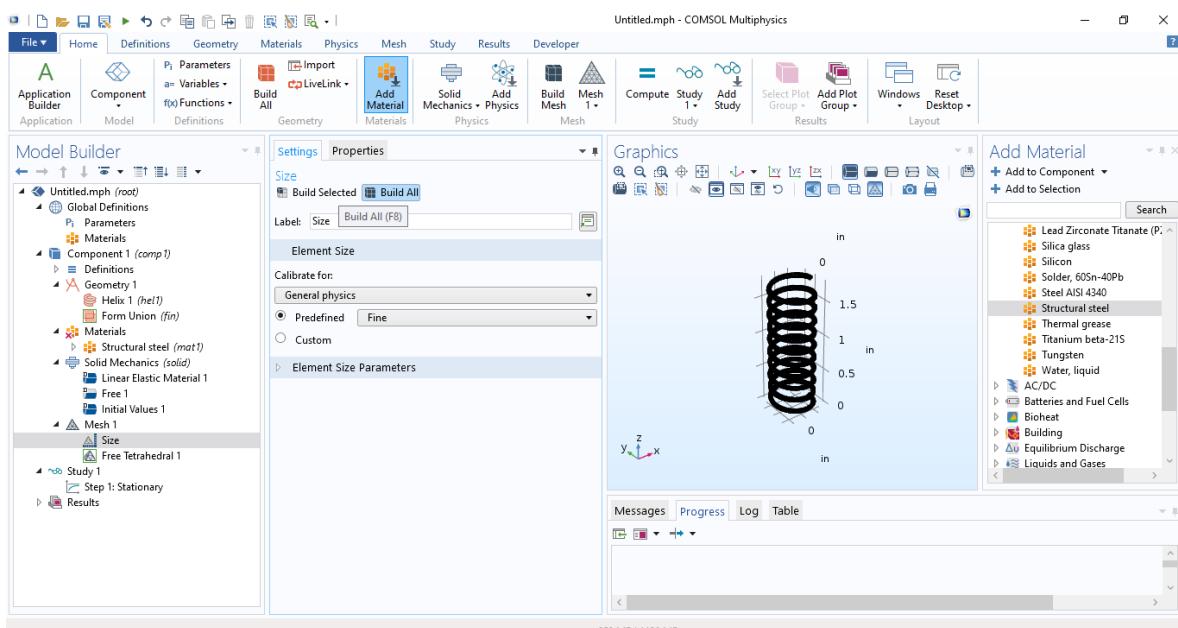
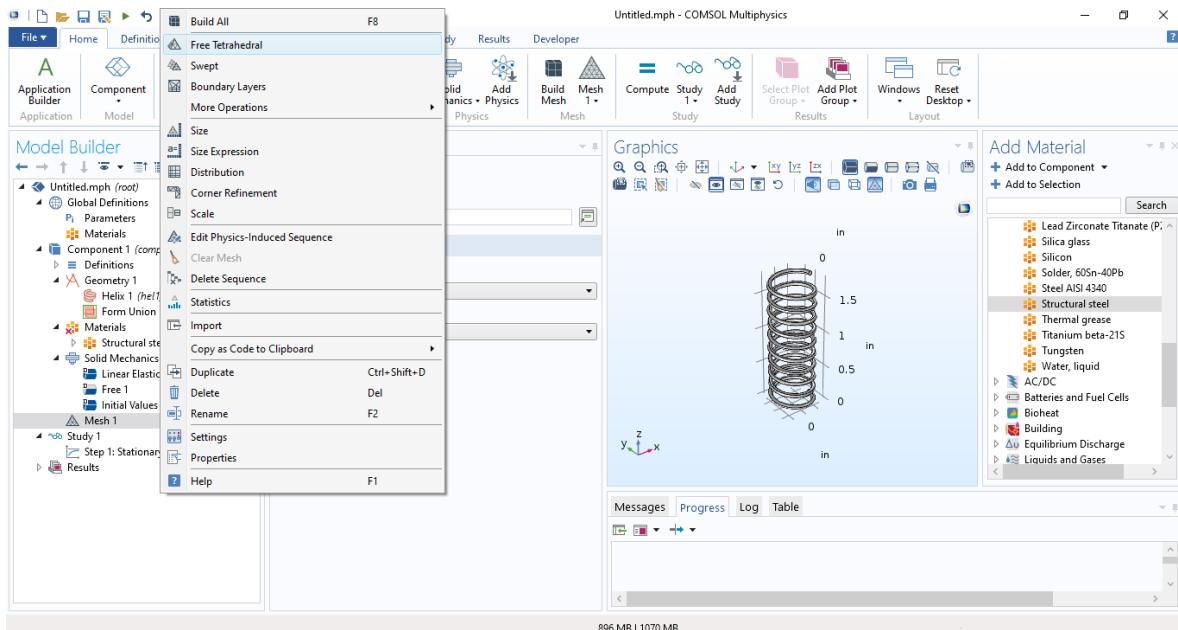


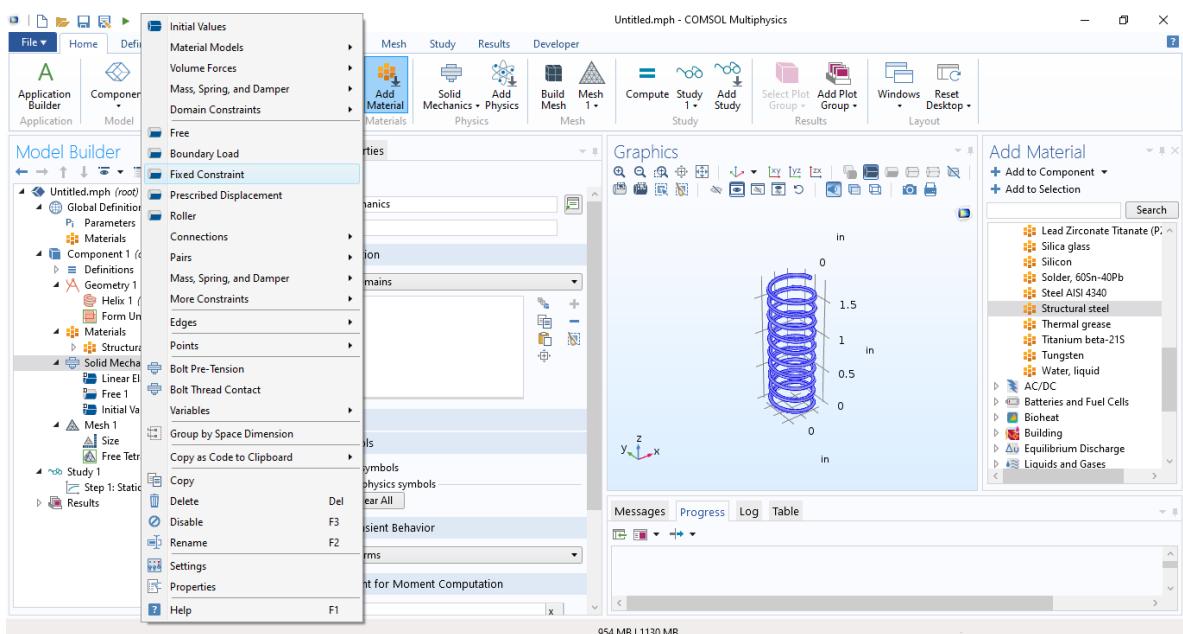
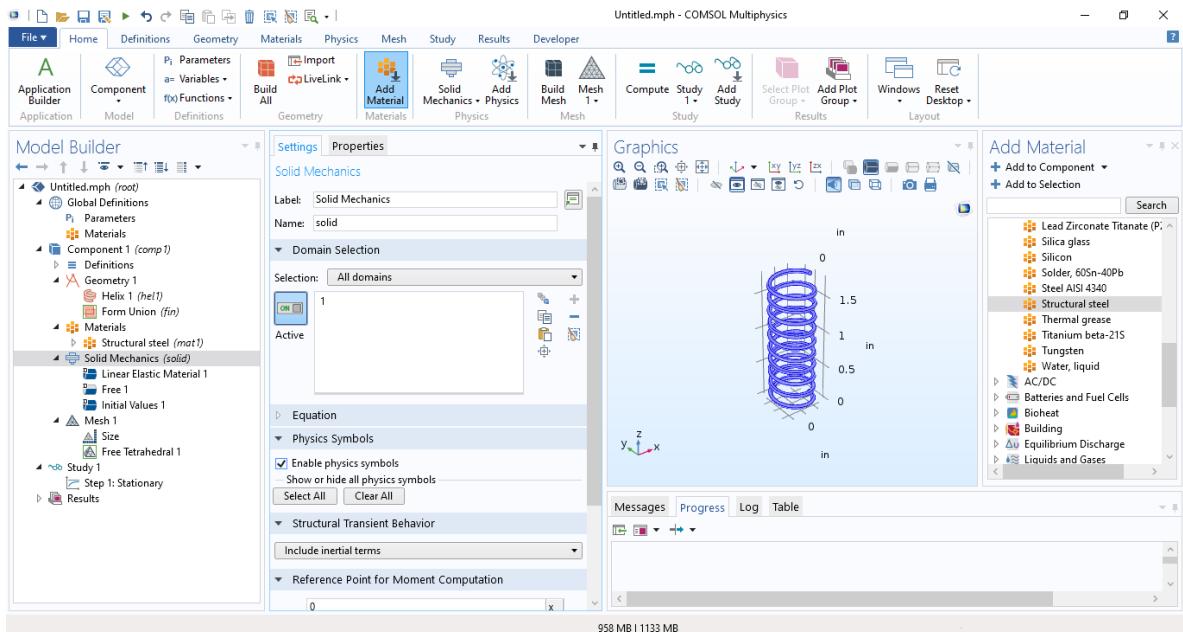
ANÁLISIS MECÁNICO EN COMSOL:

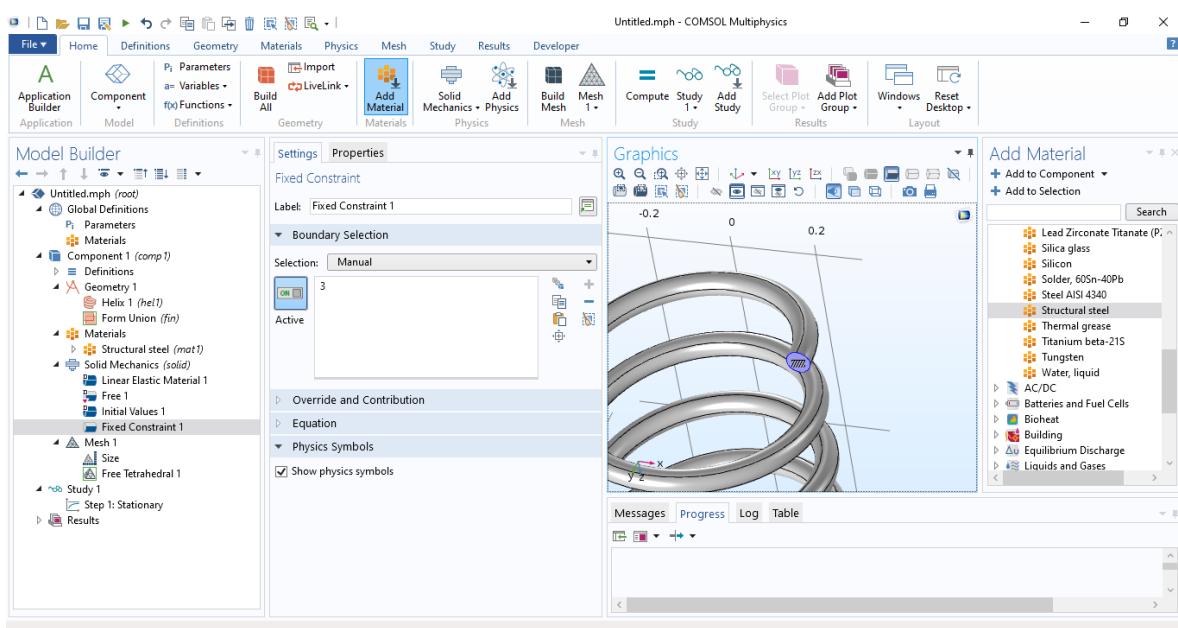
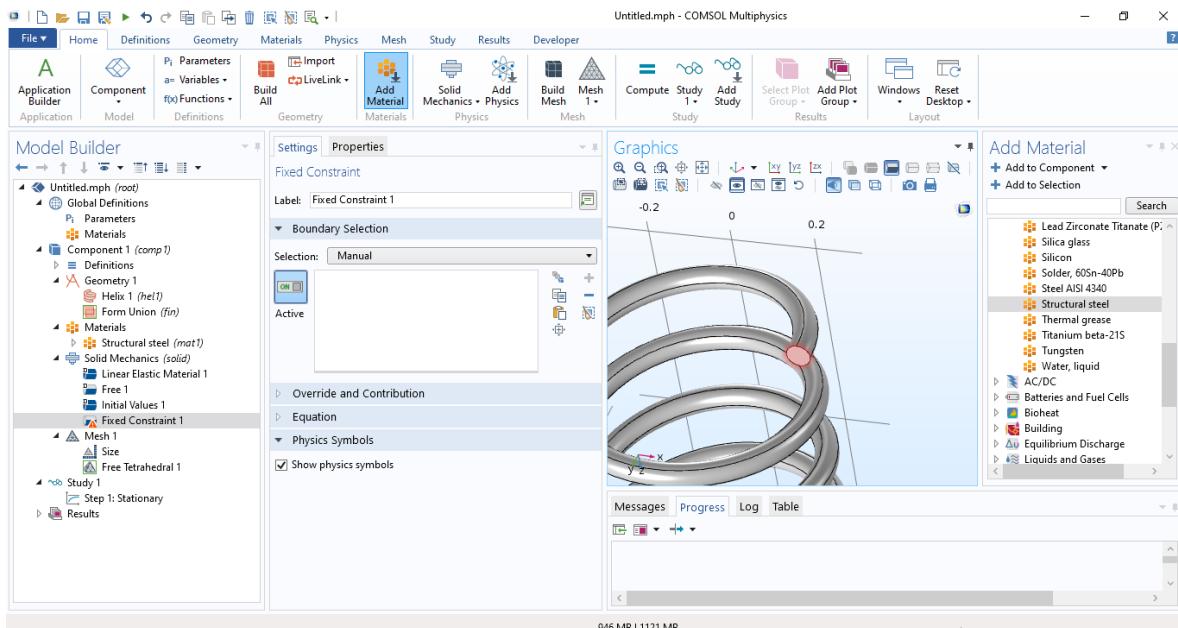
El resorte se puede torcer hacia un lado o hacia otro.

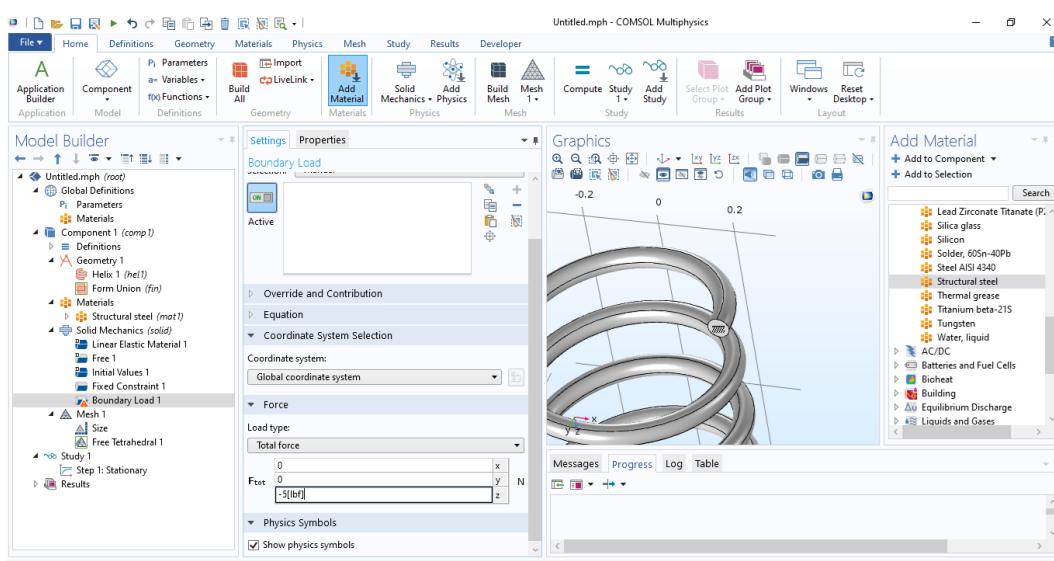
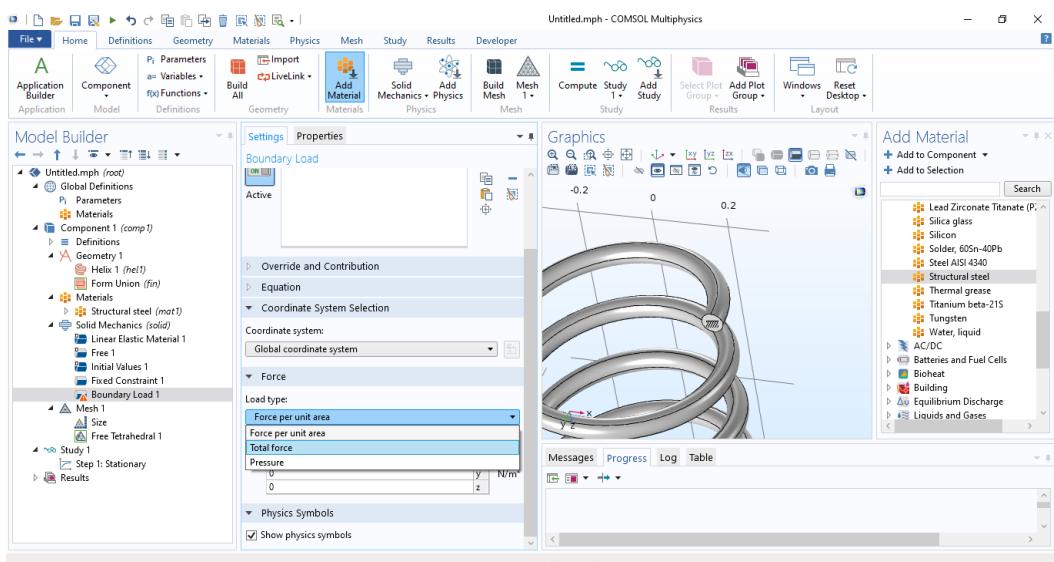
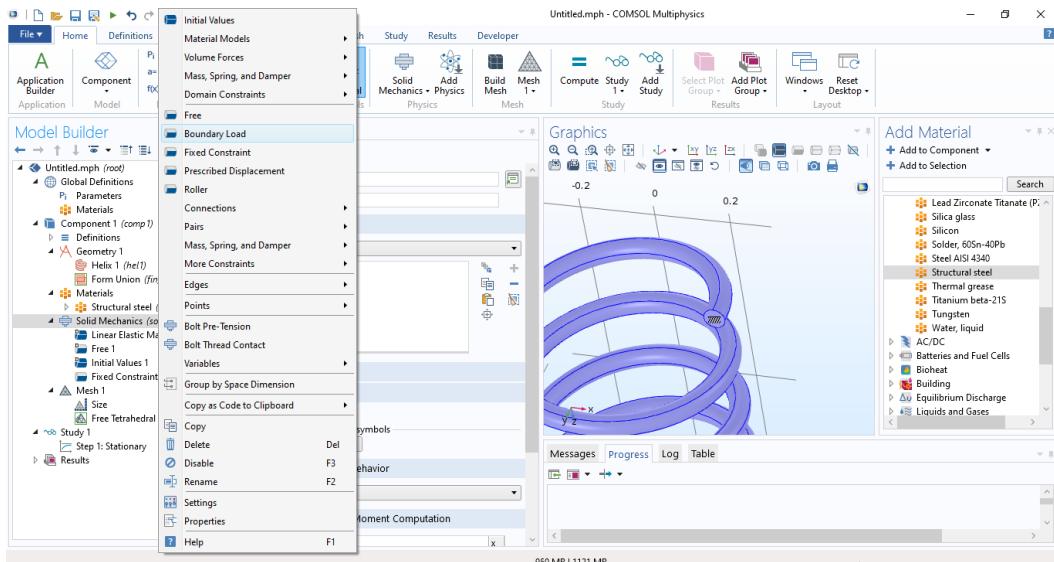




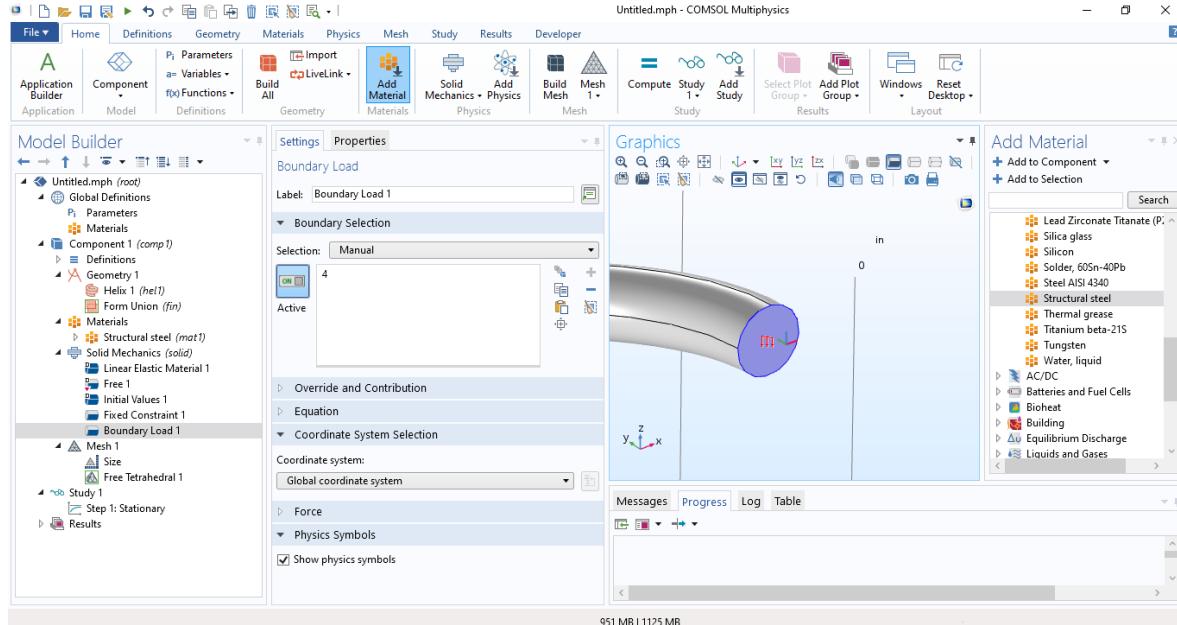




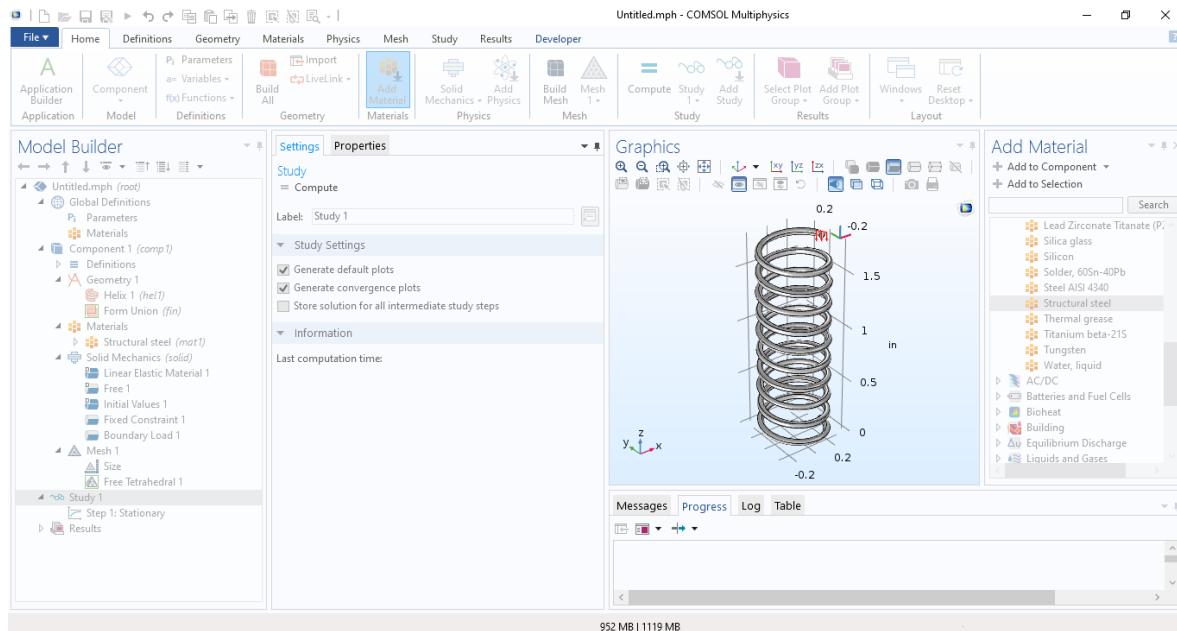




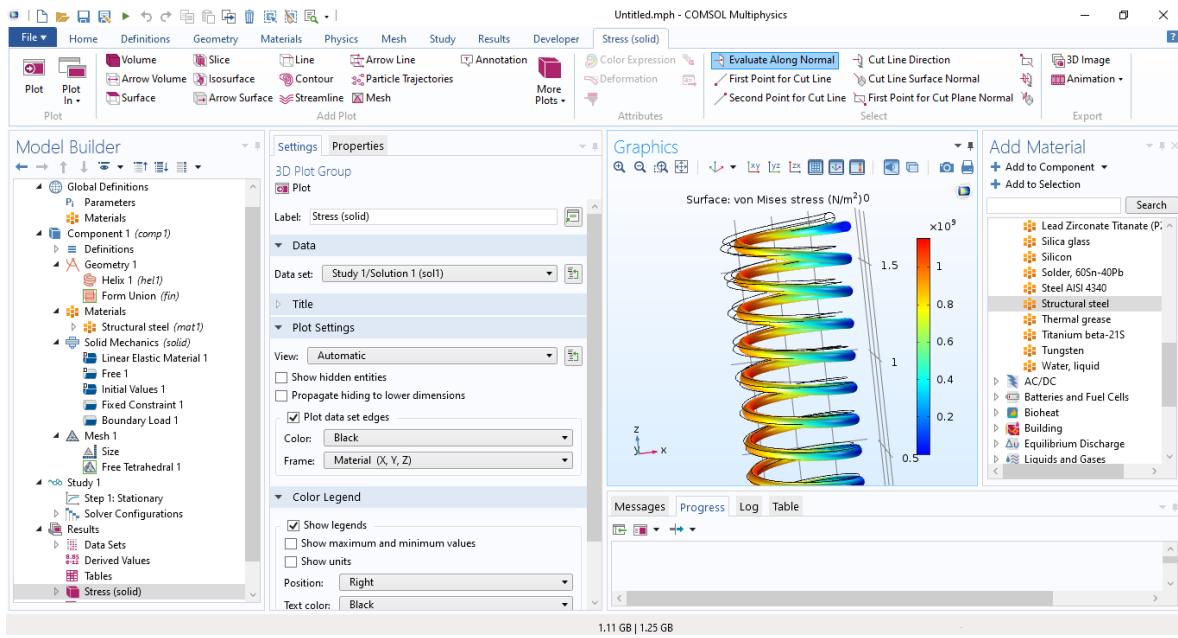
RESULTADO DEL ELEMENTO FINITO EN COMSOL:



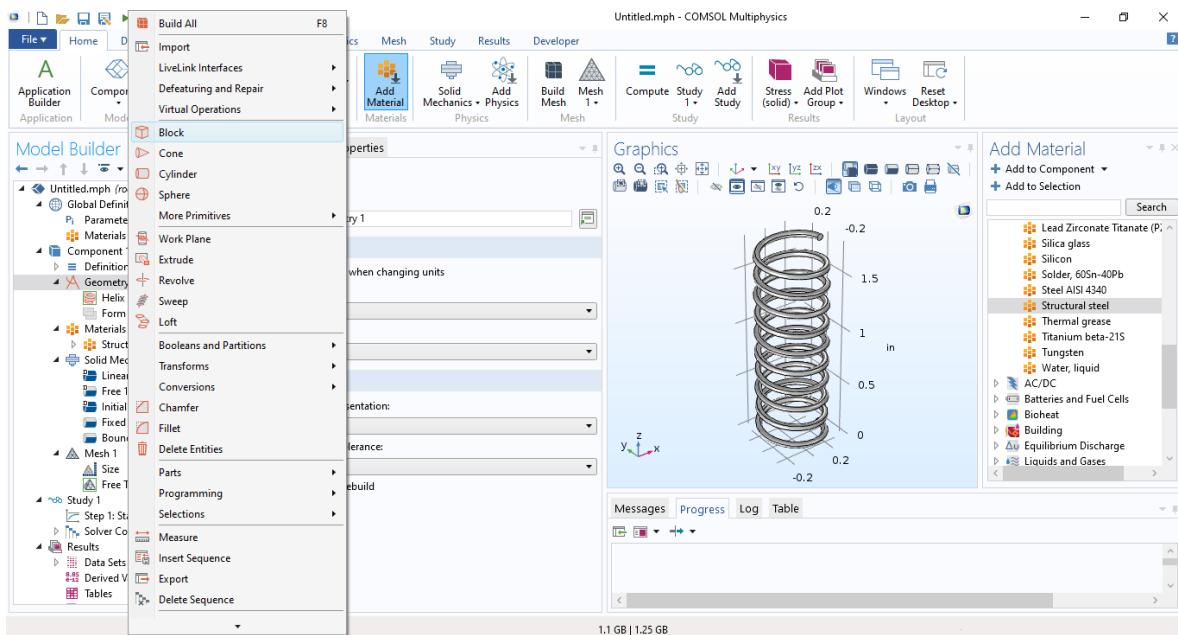
Poner el soporte de la fuerza de esta manera no funcionaría porque pandearía el resorte.



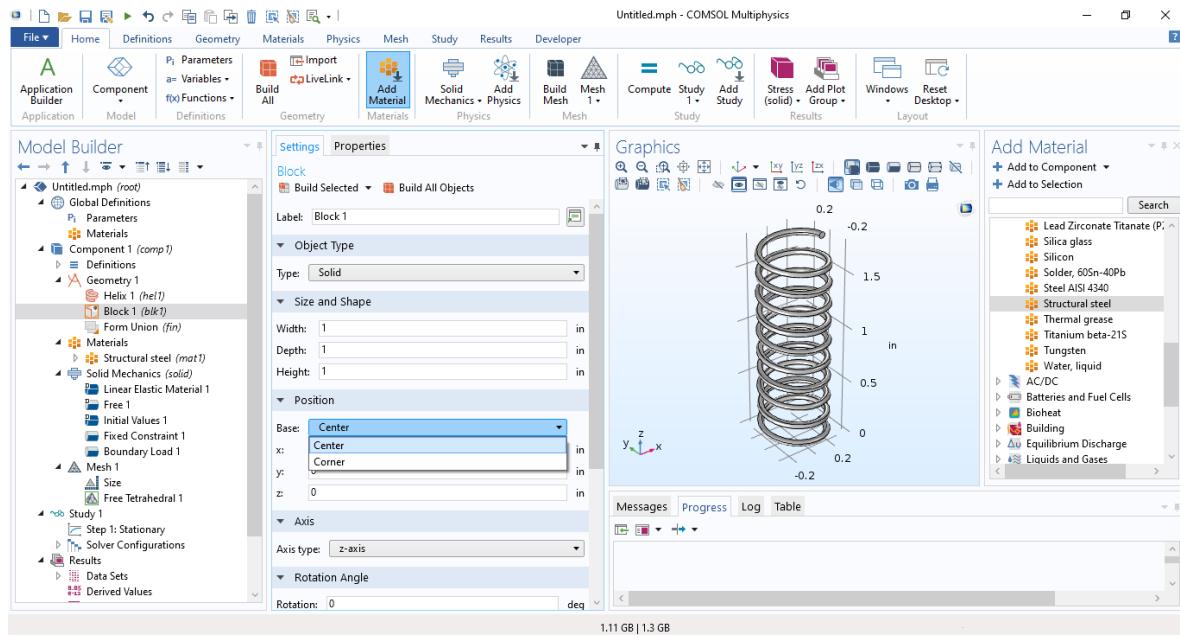
Podemos ver que el resorte se pandea hacia el lado derecho.



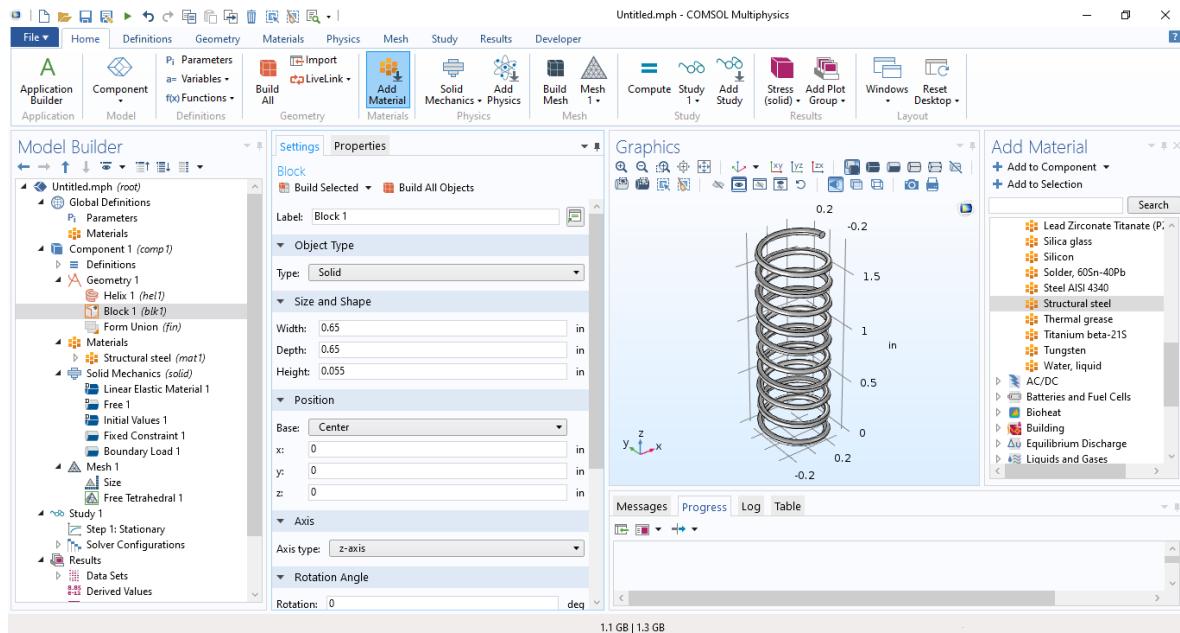
Para que podamos ver la compresión del resorte debemos agregar dos placas a ambos lados.

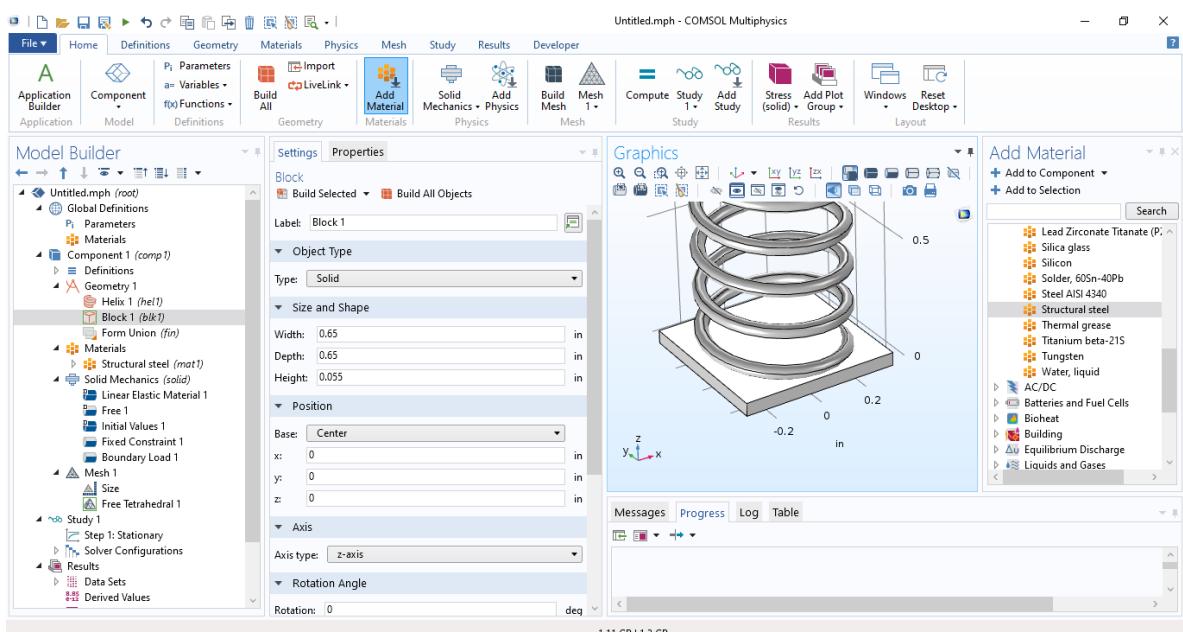
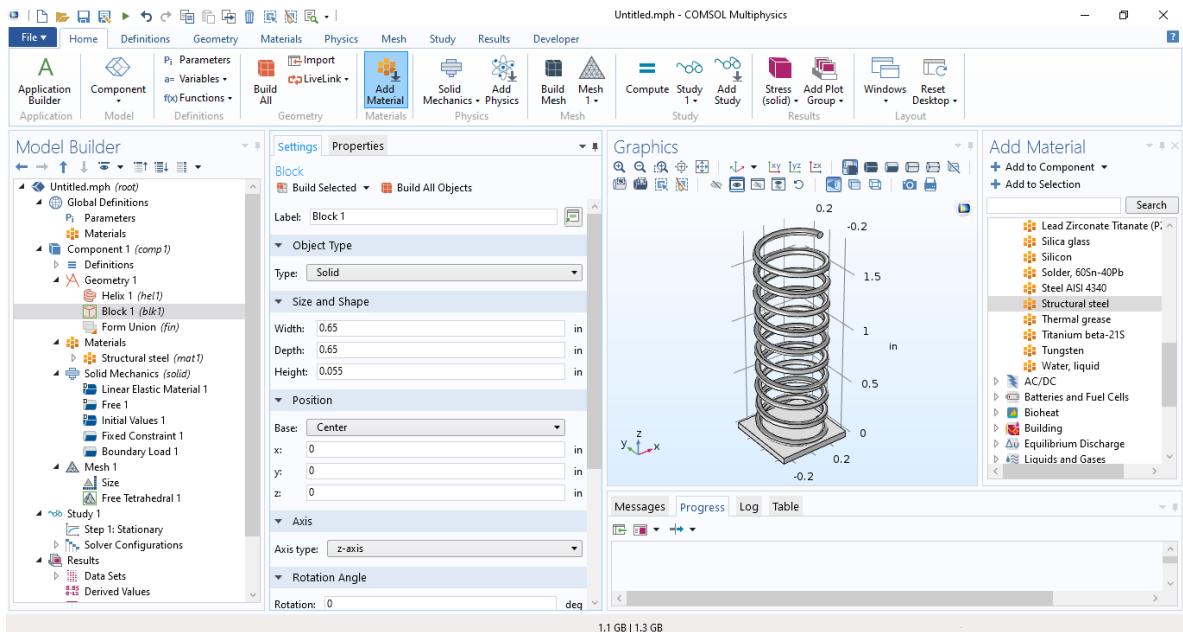


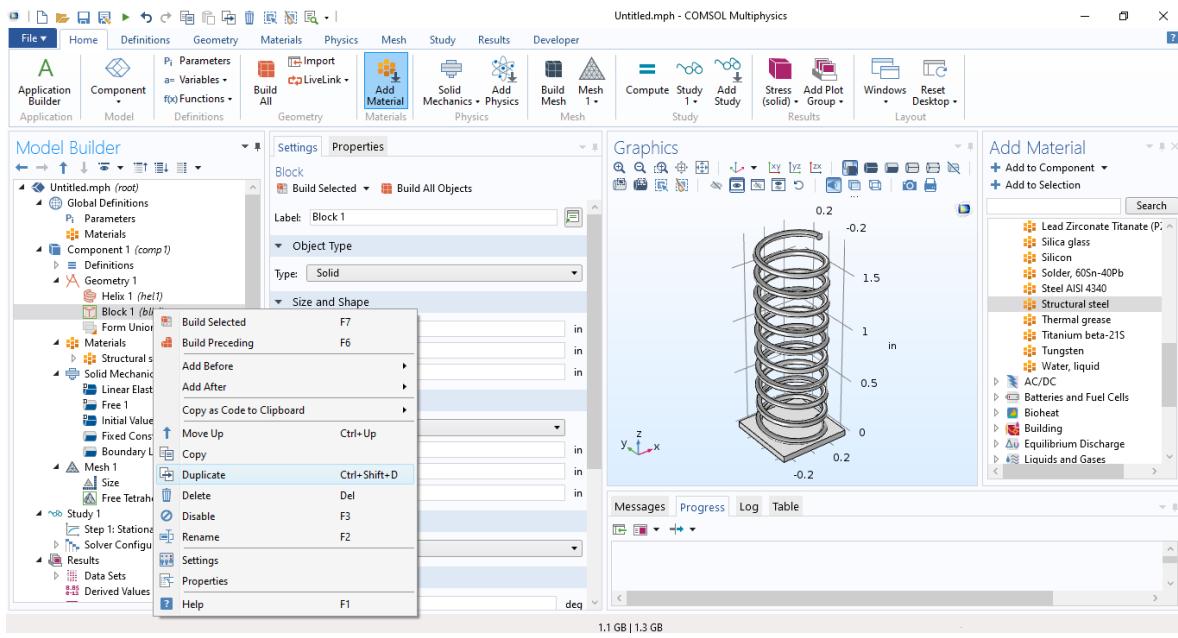
Con la opción de Block → Position → Base: → Center, le indico al programa que el centro de las placas empiece en el centro del círculo del resorte.



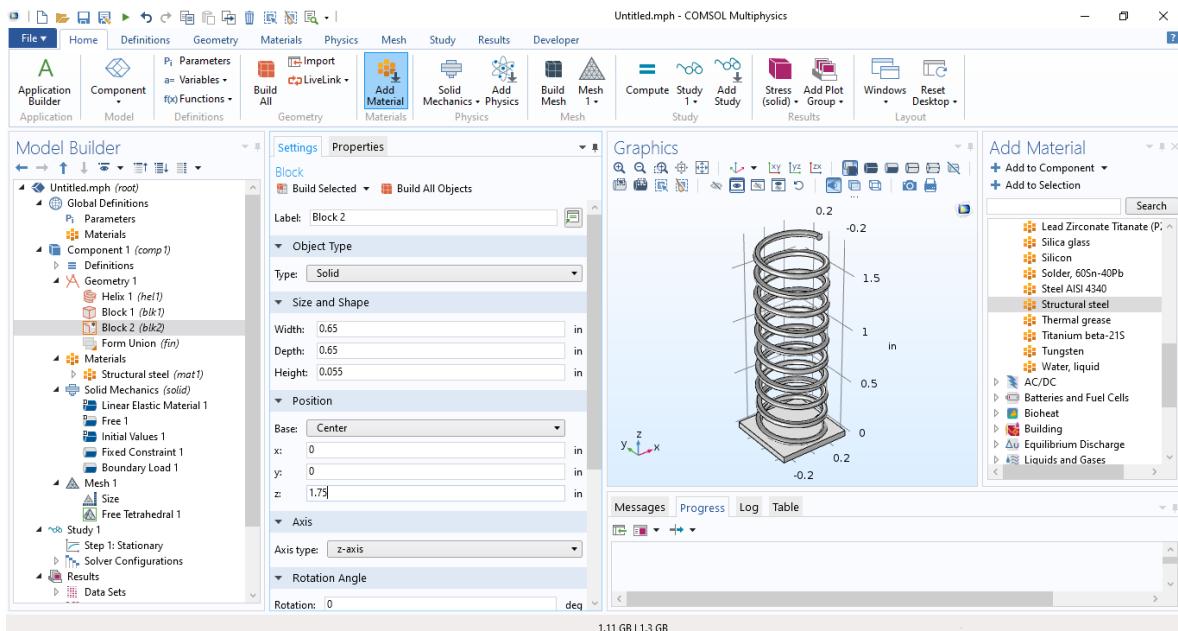
El ancho de la placa pongo que sea el mismo que el diámetro del alambre.

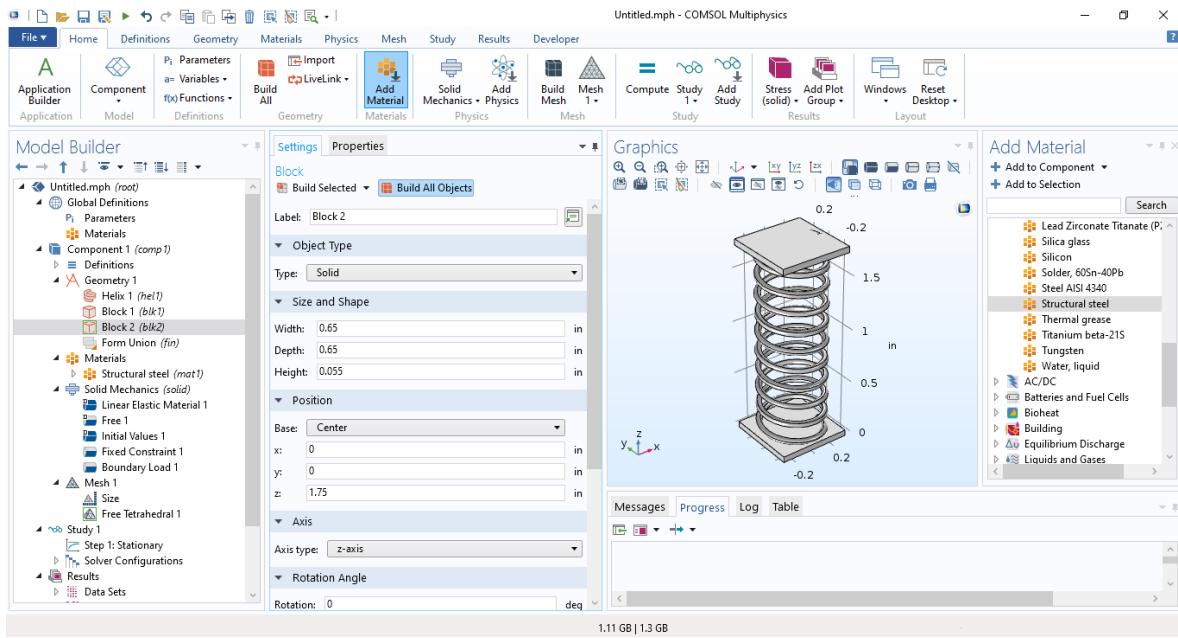




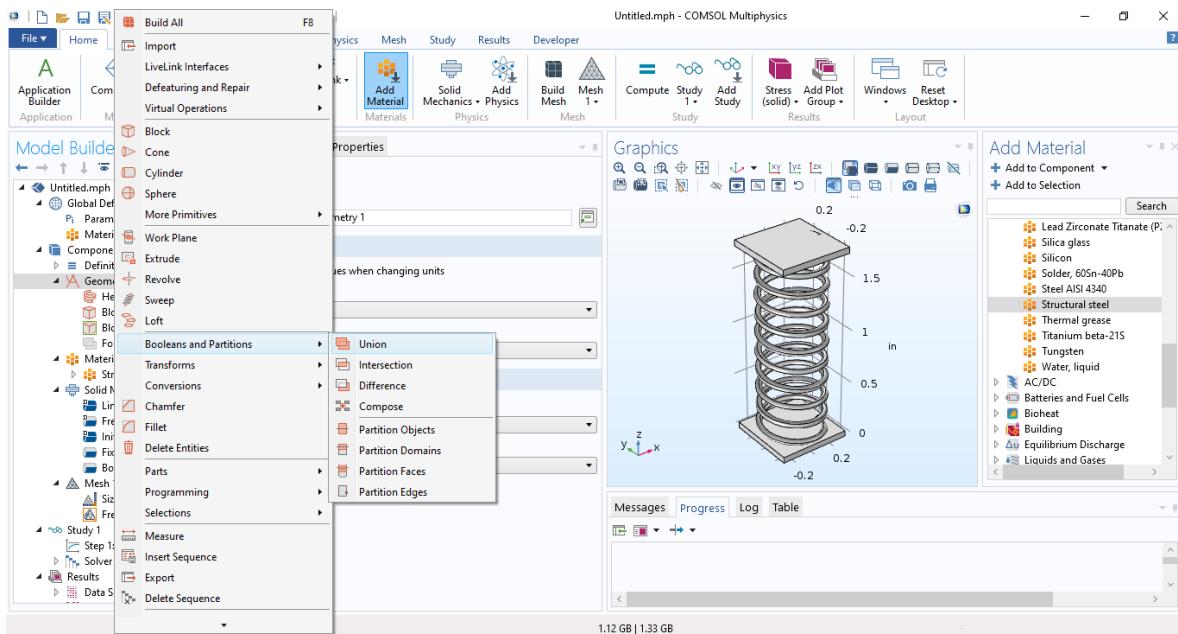


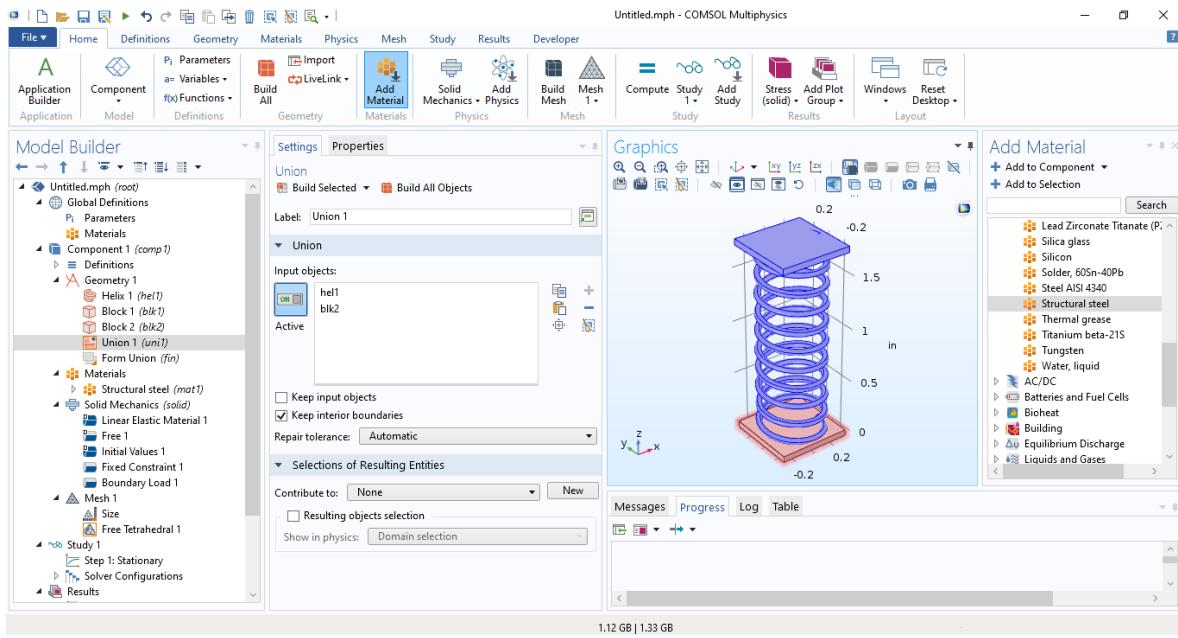
Ahora copio la placa y la pongo en una ubicación Z que sea igual al valor de Lf.



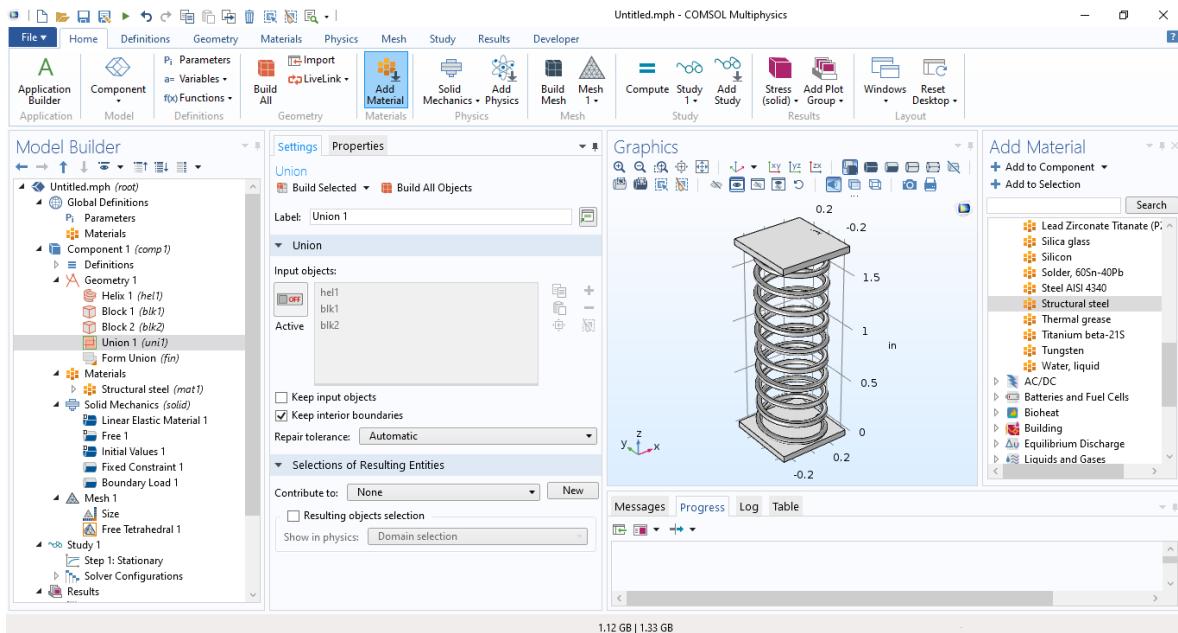


En este momento las placas son independientes al resorte, no un solo volumen, por lo tanto, las debo unir de la siguiente manera:

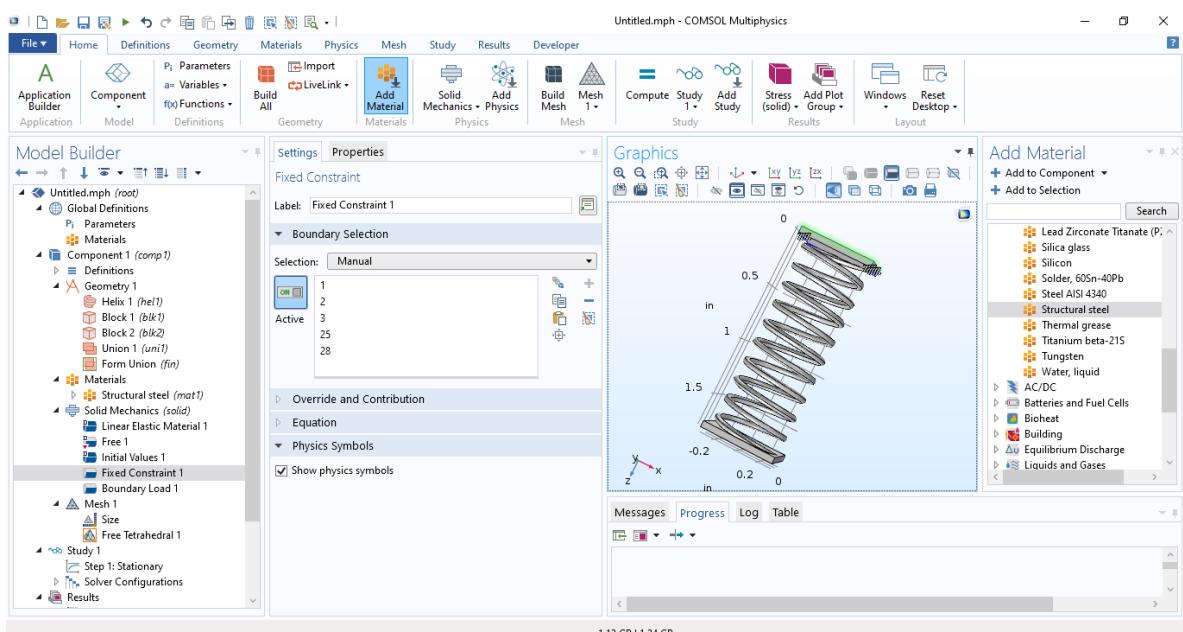
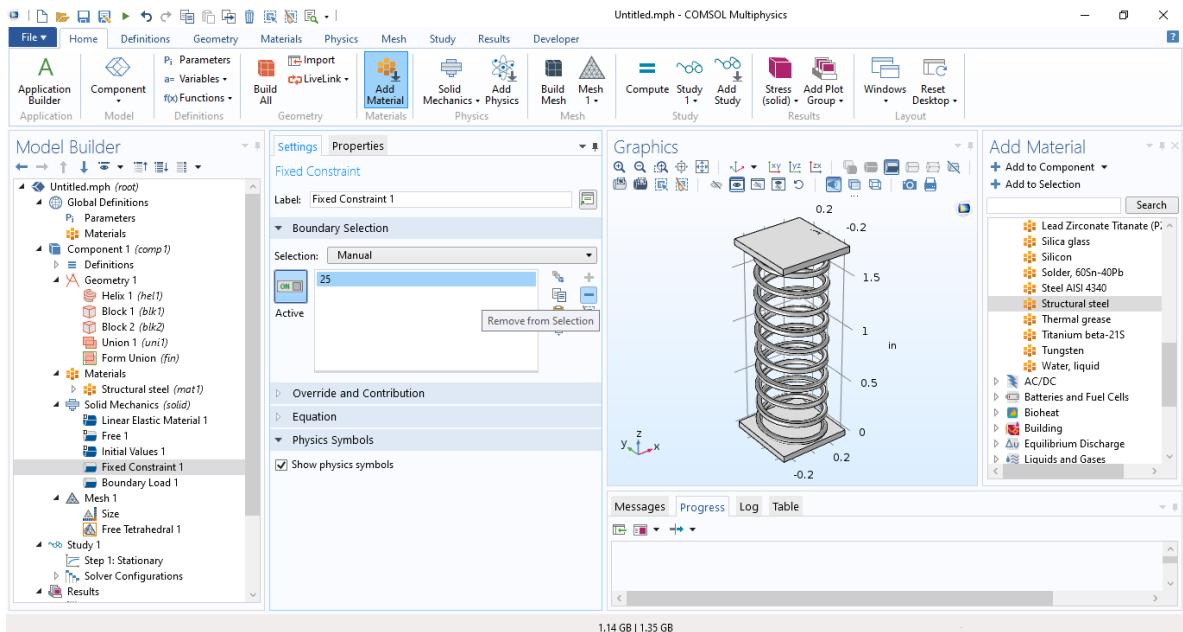


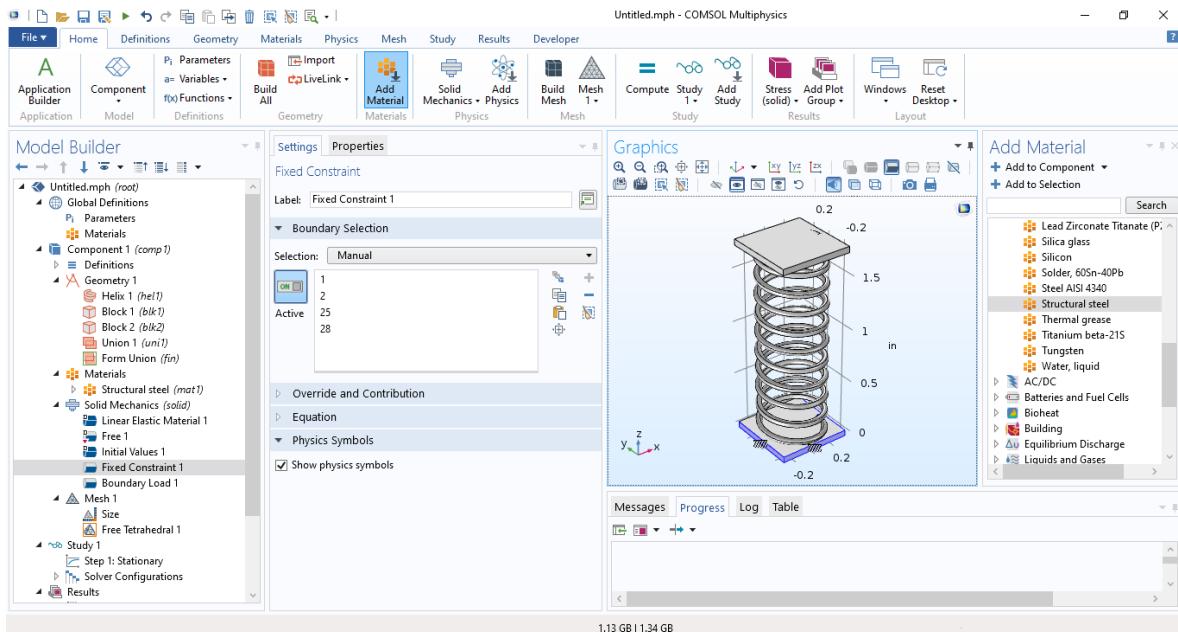


Con ello ya se unen todos mis elementos en uno solo.

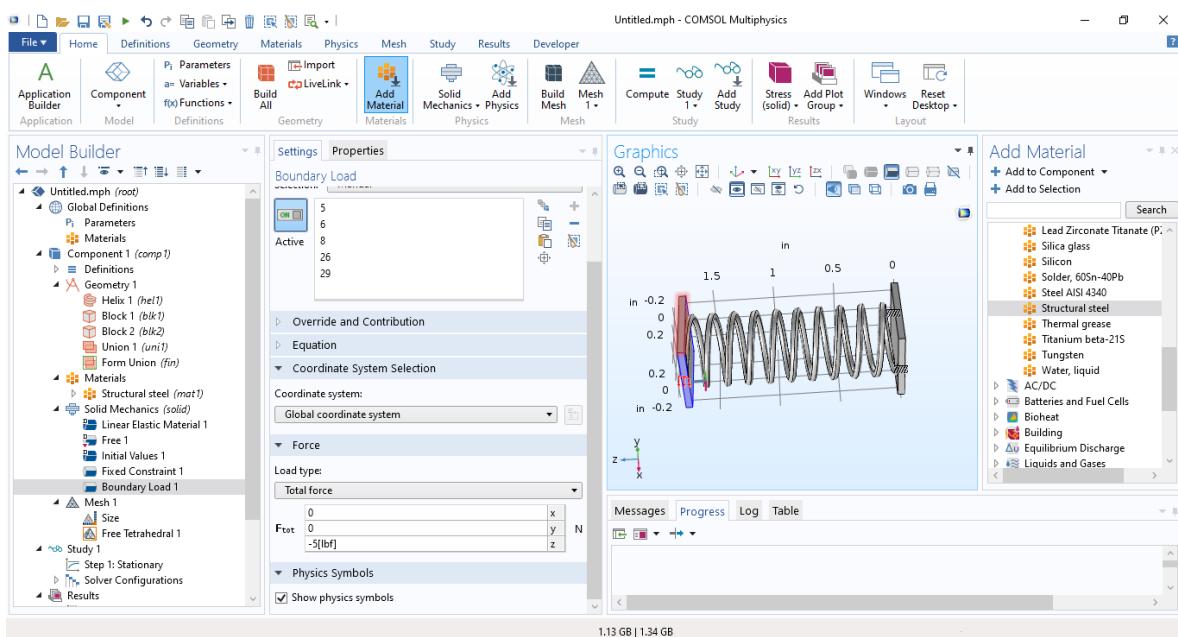


Dejo lo que había seleccionado en fixed constraint (el apoyo) y restrinjo todas las áreas de la placa para poder observar el efecto de la compresión sobre el resorte.

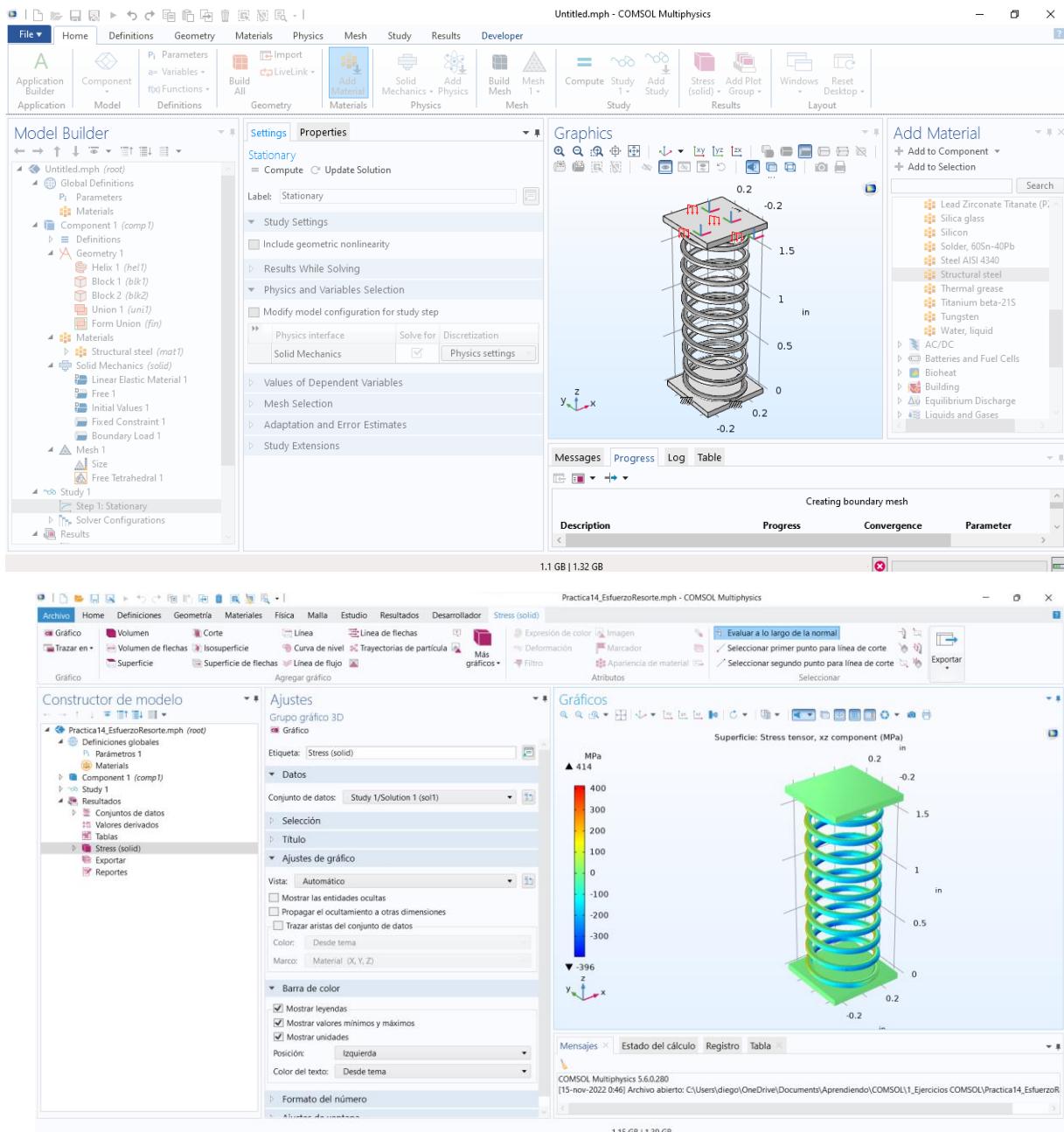




Y en boundary load (la carga externa) hago lo mismo.



Y ahora si ya realizo el estudio mecánico de nuevo para observar la compresión del resorte.



BIBLIOGRAFÍA:

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