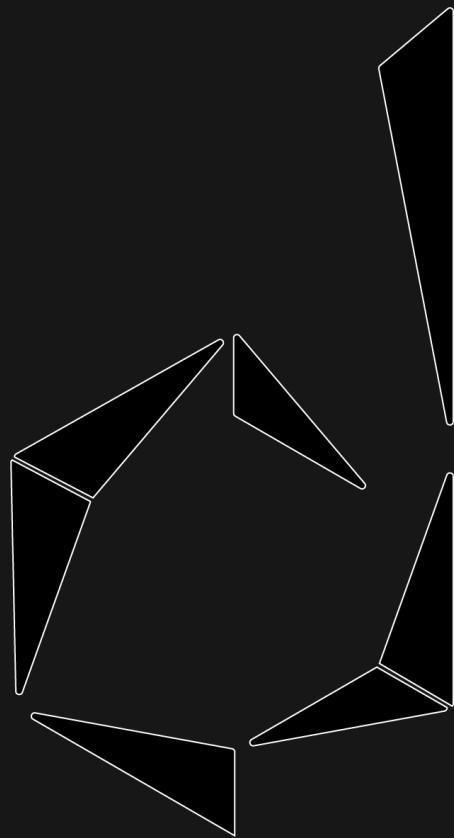


# INGENIERÍA MECATRÓNICA



## DI\_CERO

DIEGO CERVANTES RODRÍGUEZ

INGENIERÍA ASISTIDA POR COMPUTADORA

COMSOL MULTIPHYSICS 5.6

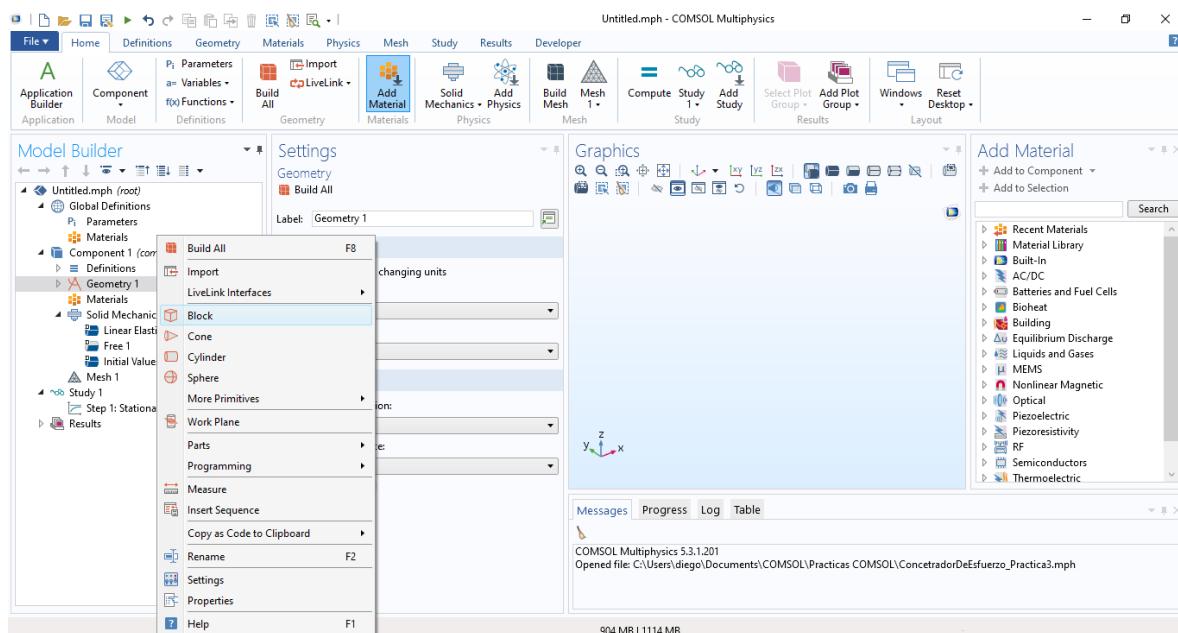
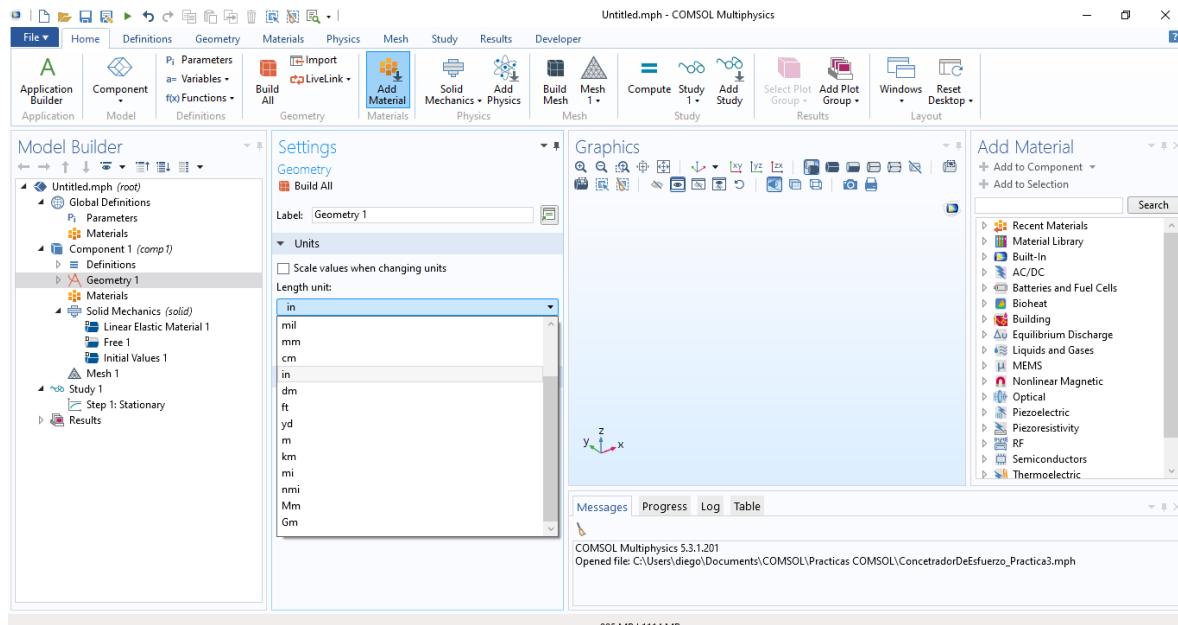
3.1: Placa 3D con Parte Flaca por  
Barrenos Laterales - Tensión/Compresión

## Contenido

CREACIÓN DE LA PIEZA EN COMSOL:.....	2
ANÁLISIS MECÁNICO EN COMSOL: .....	15
RESULTADO DEL ELEMENTO FINITO EN COMSOL: .....	24
BIBLIOGRAFÍA:.....	36



# CREACIÓN DE LA PIEZA EN COMSOL:

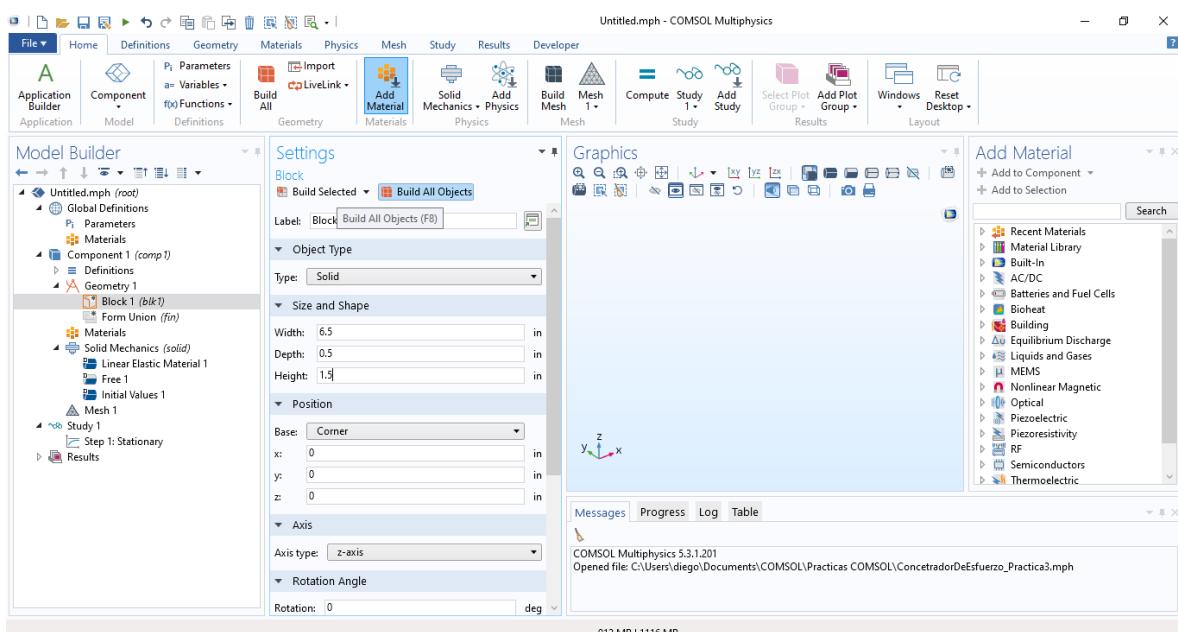
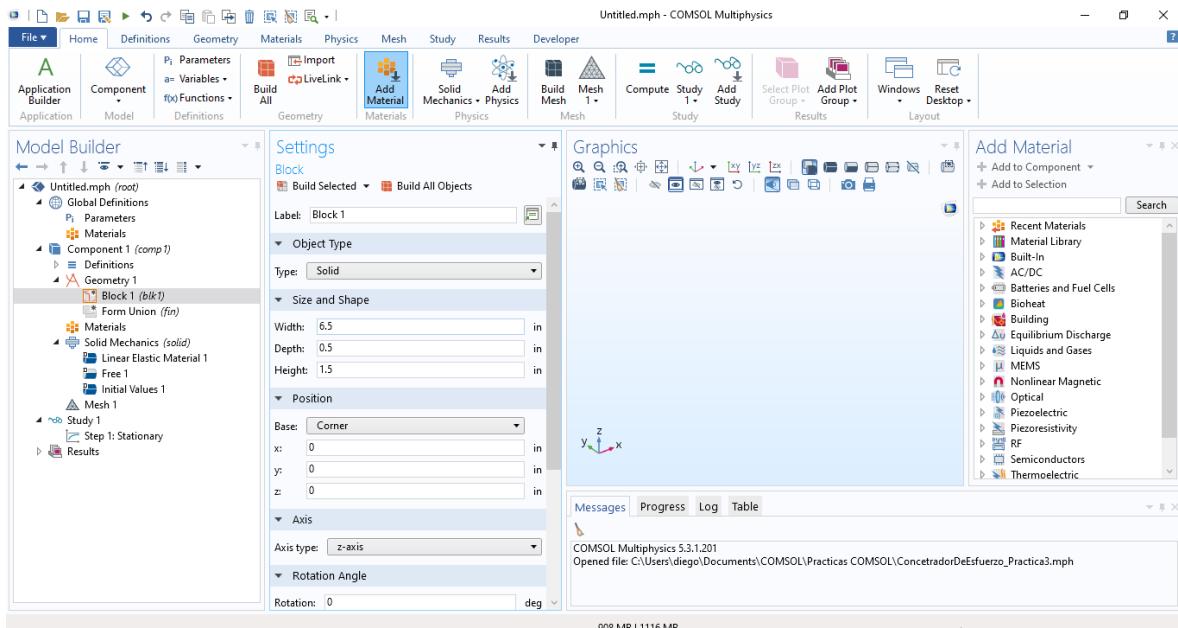


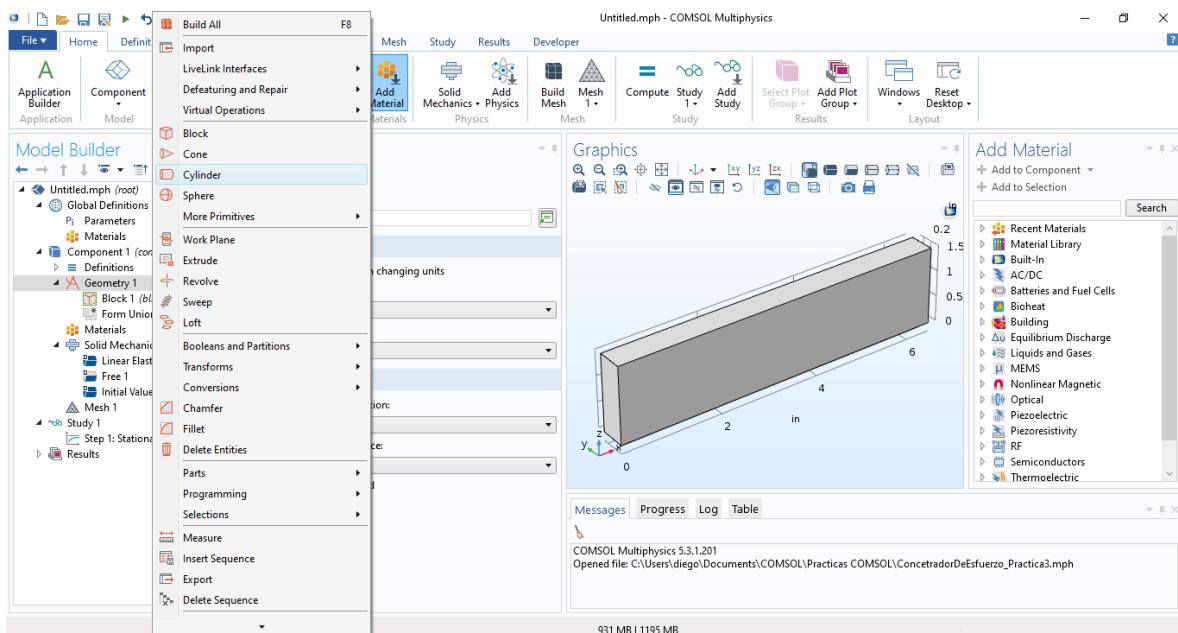
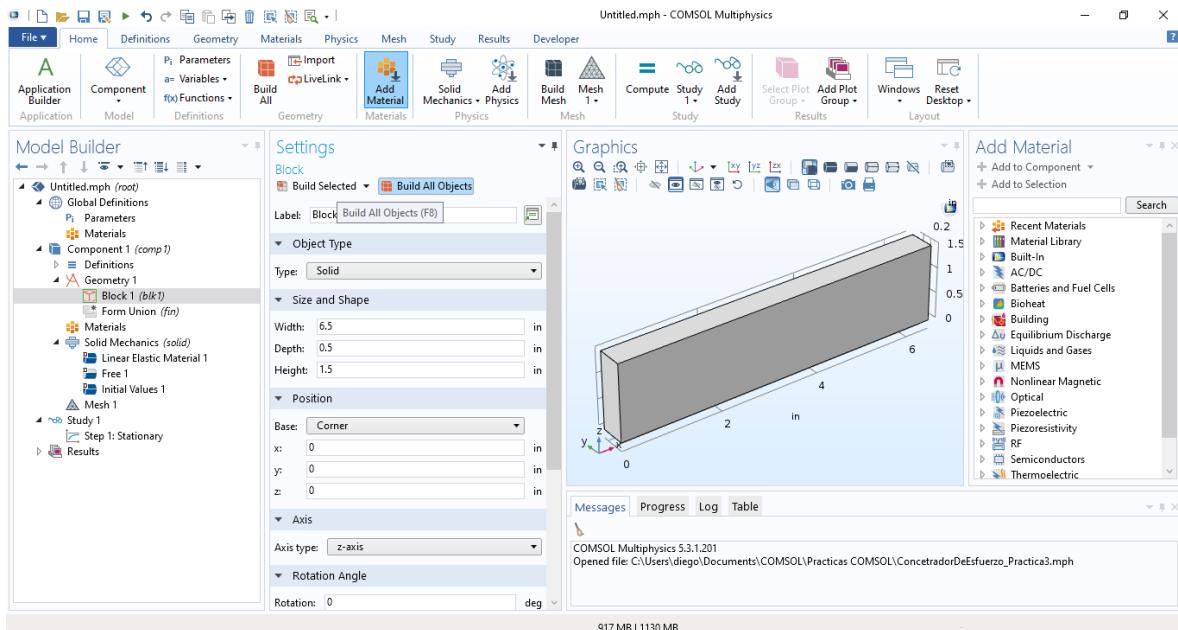
Width: x

Depth: y

Height: z

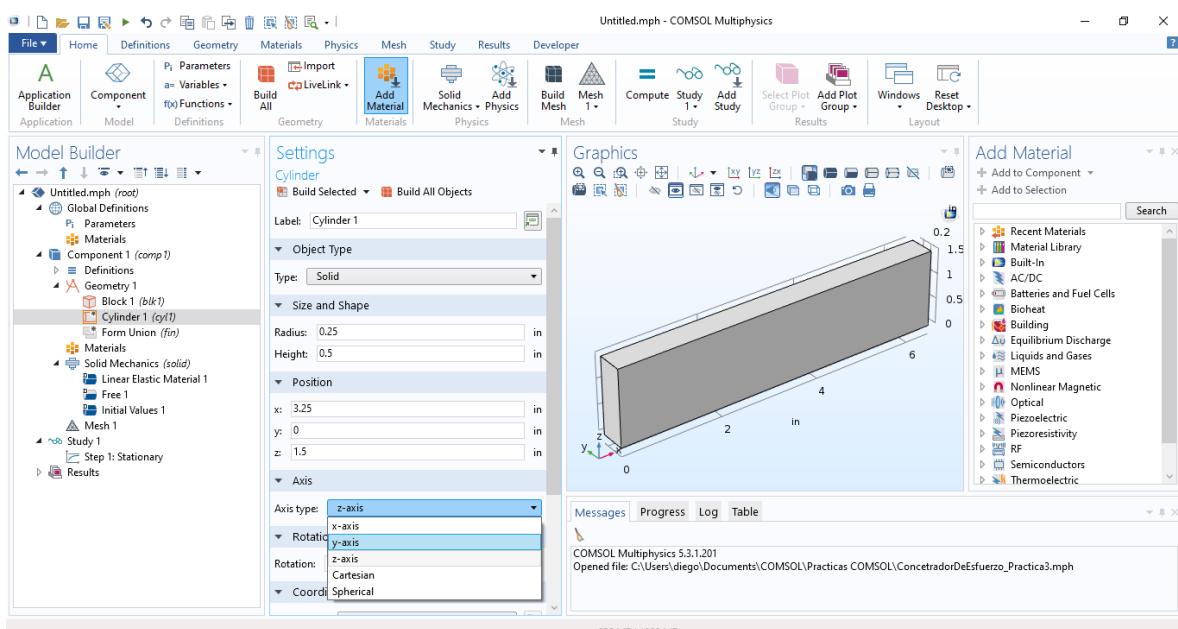
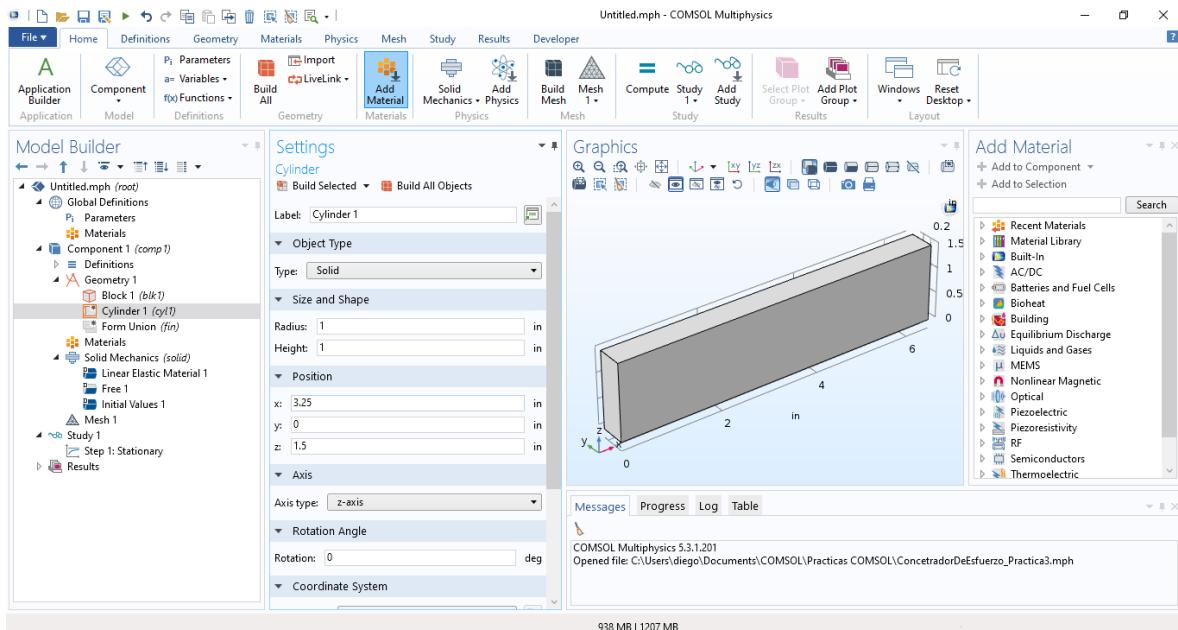


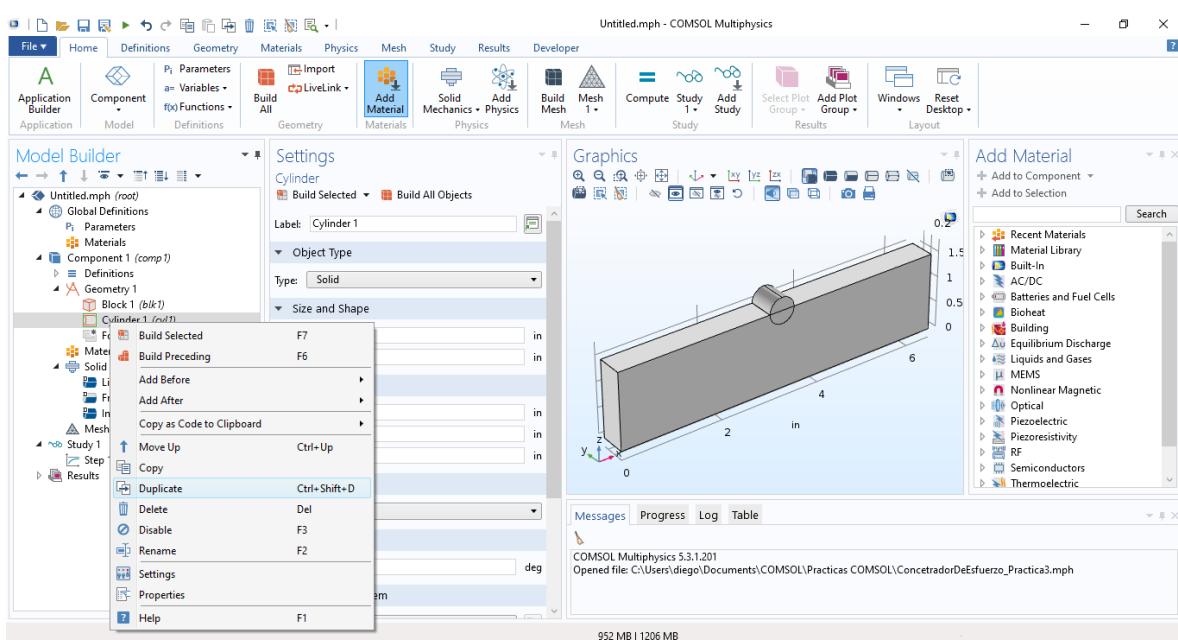
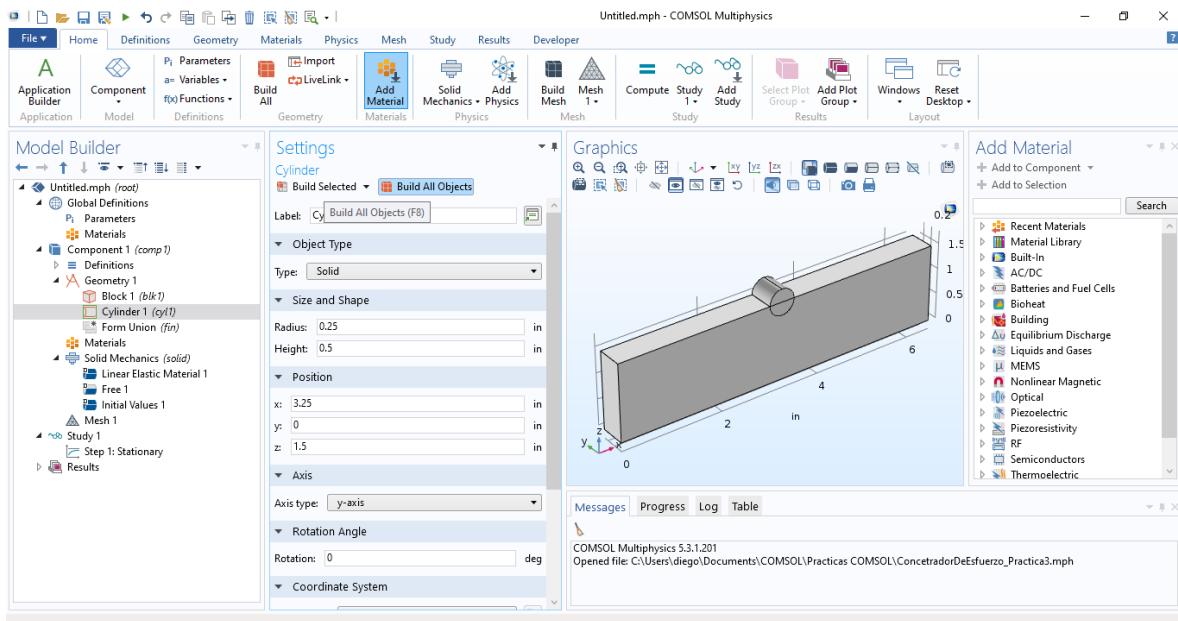


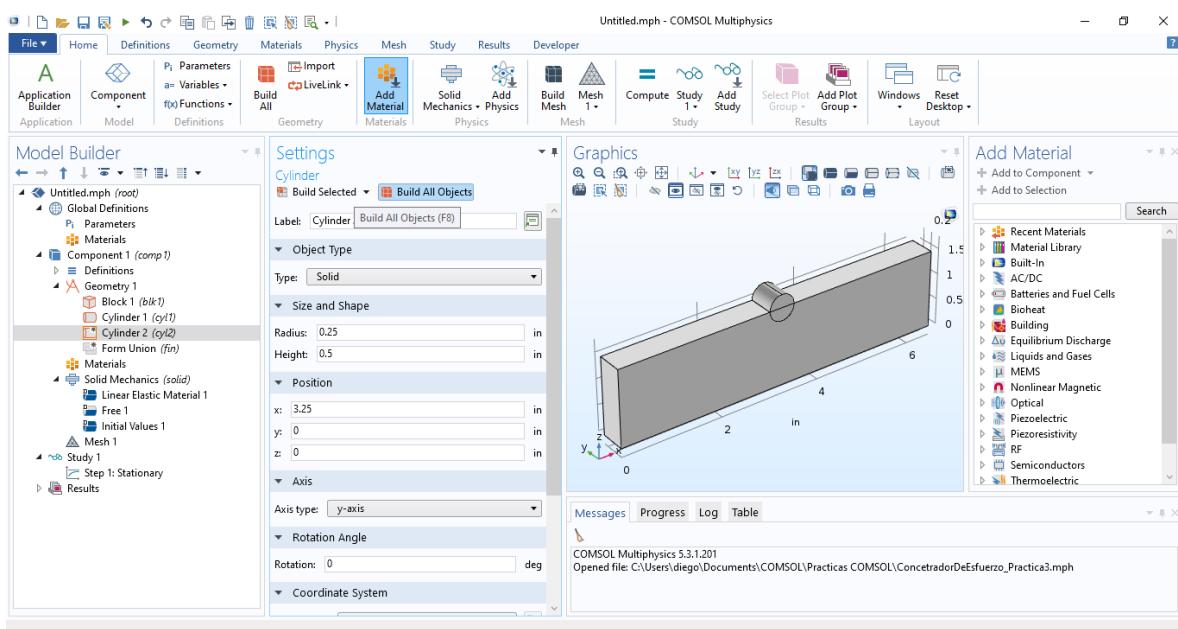
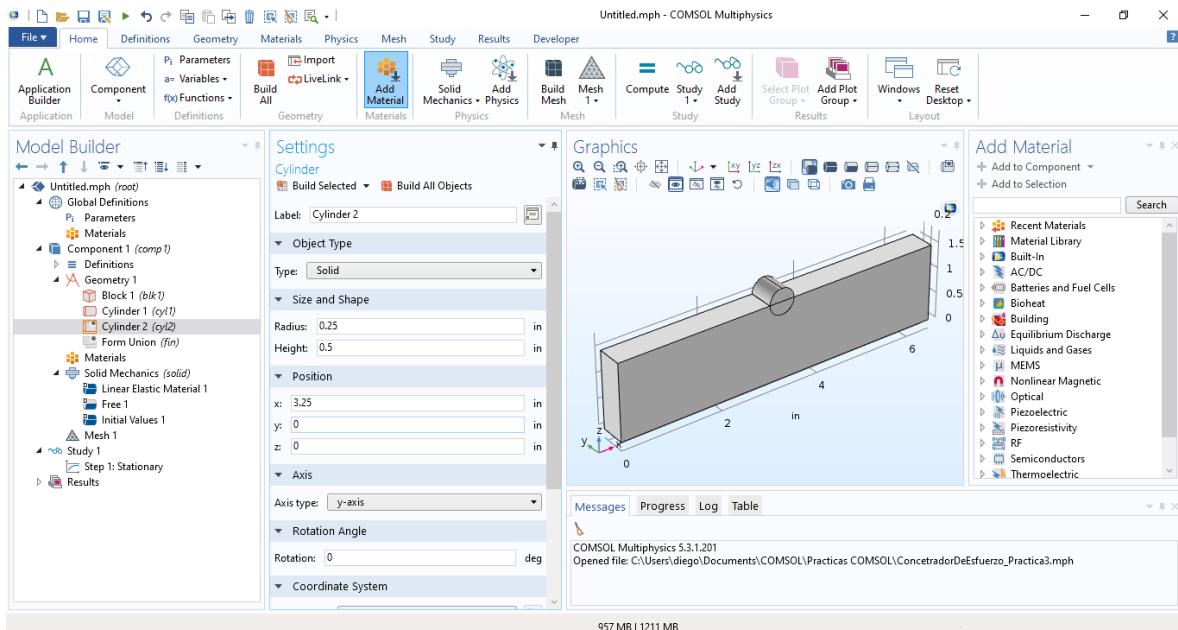


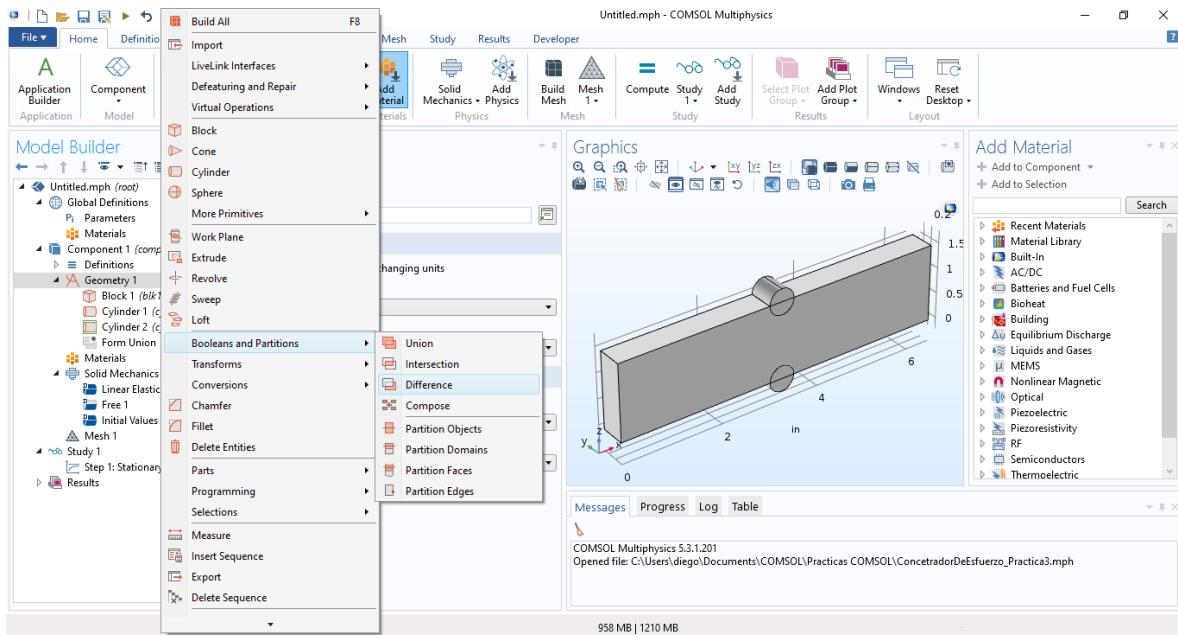
Poner el cilindro



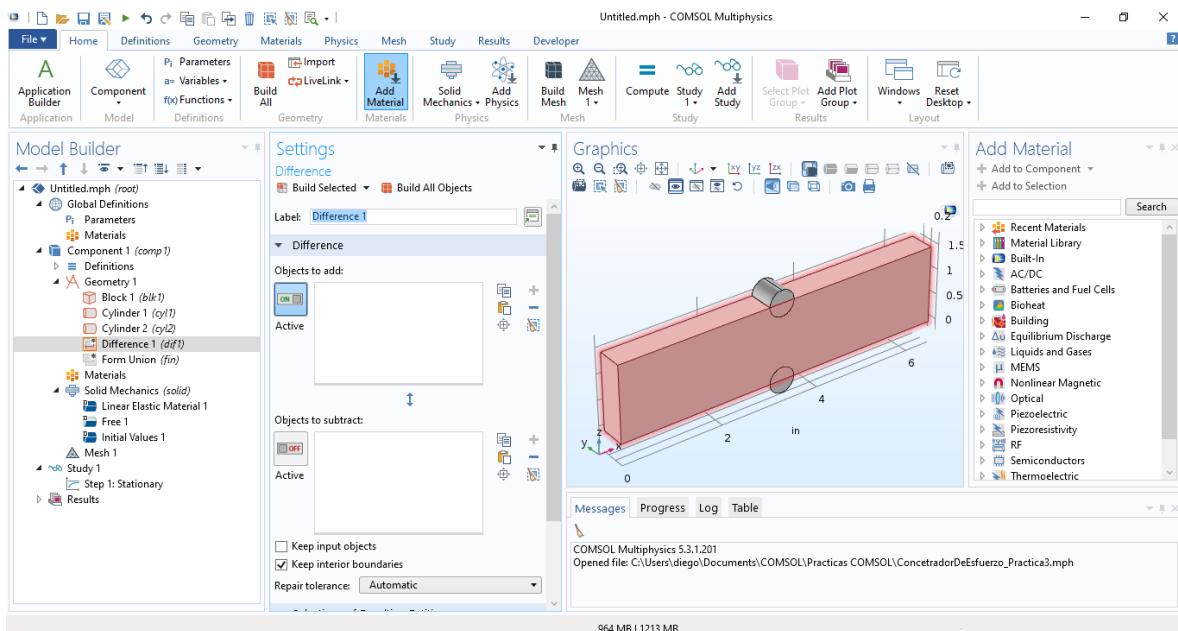


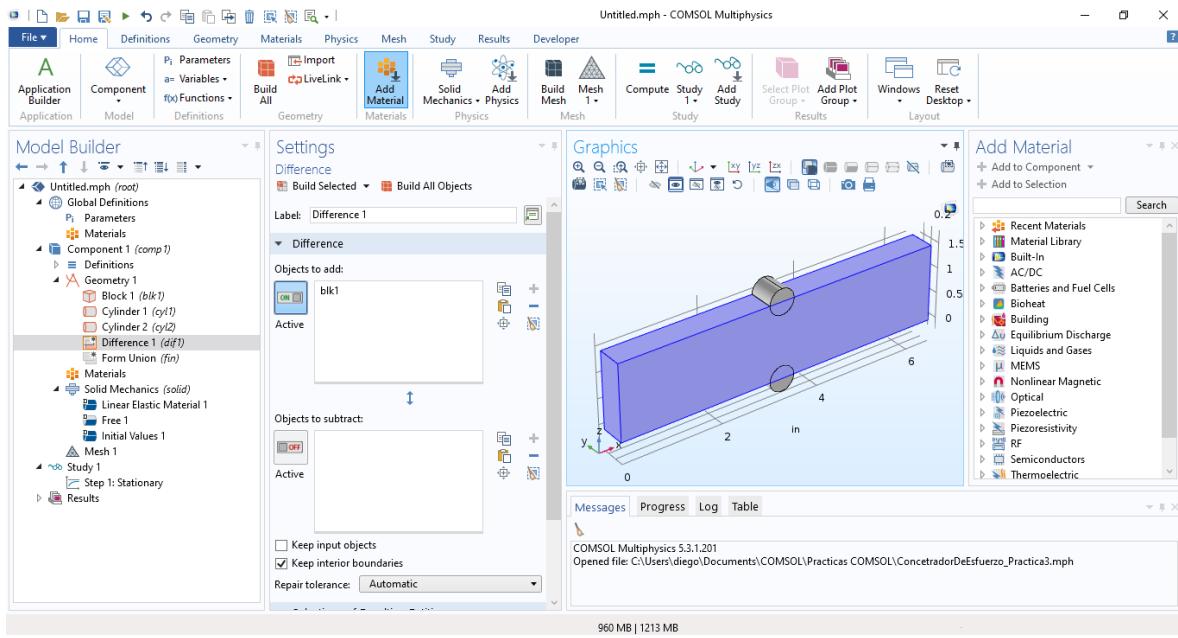




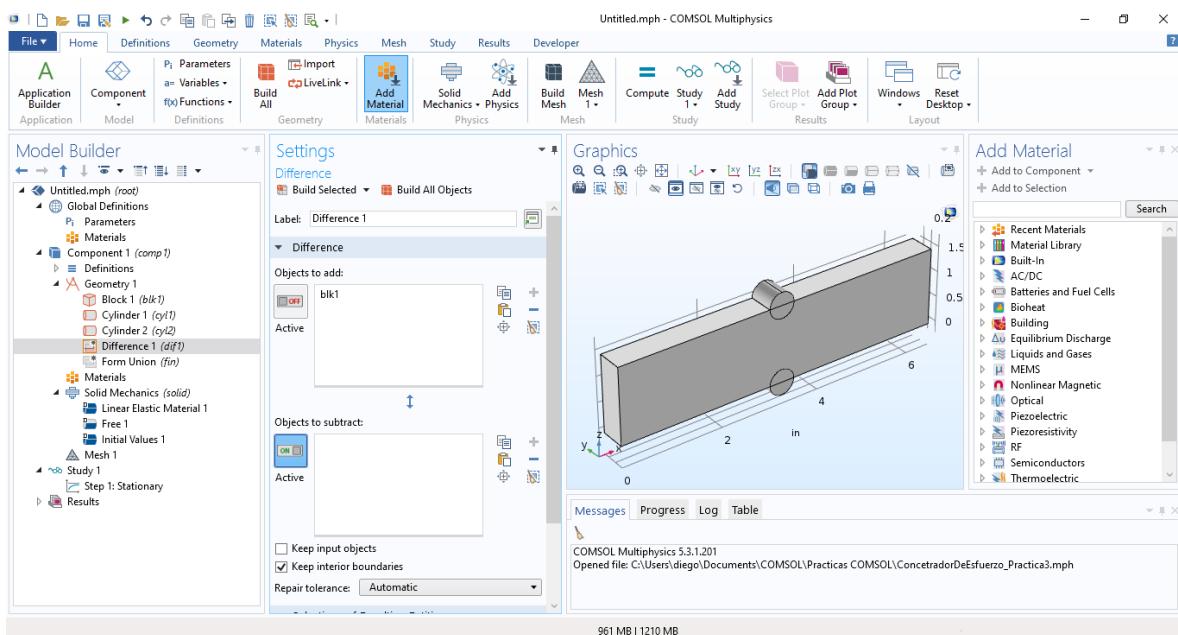


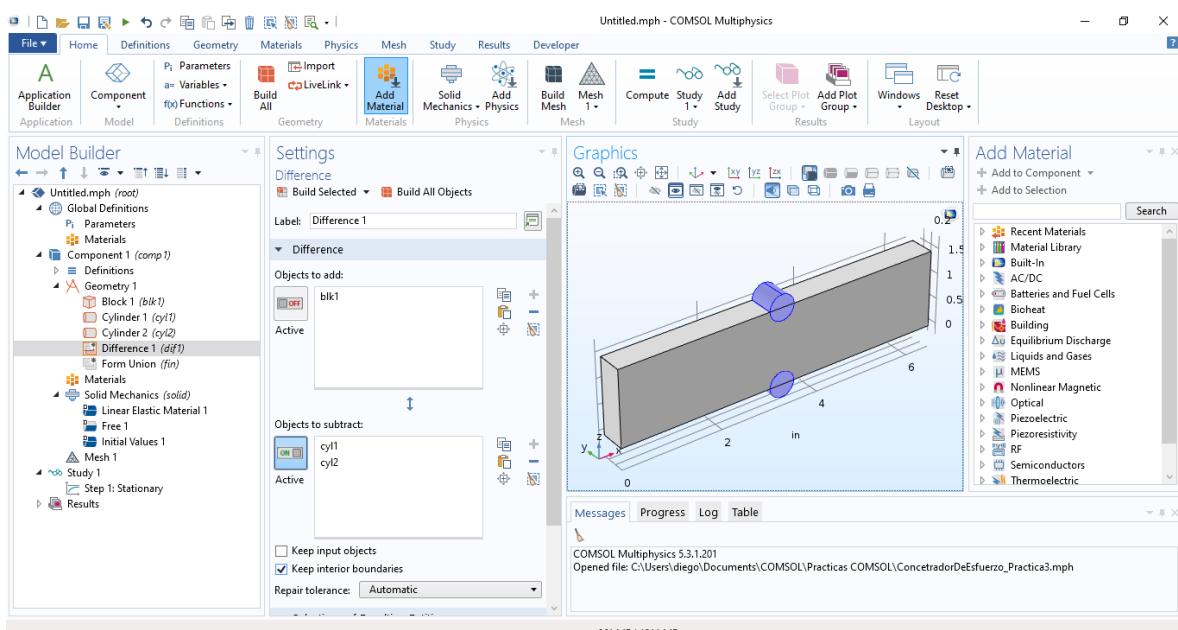
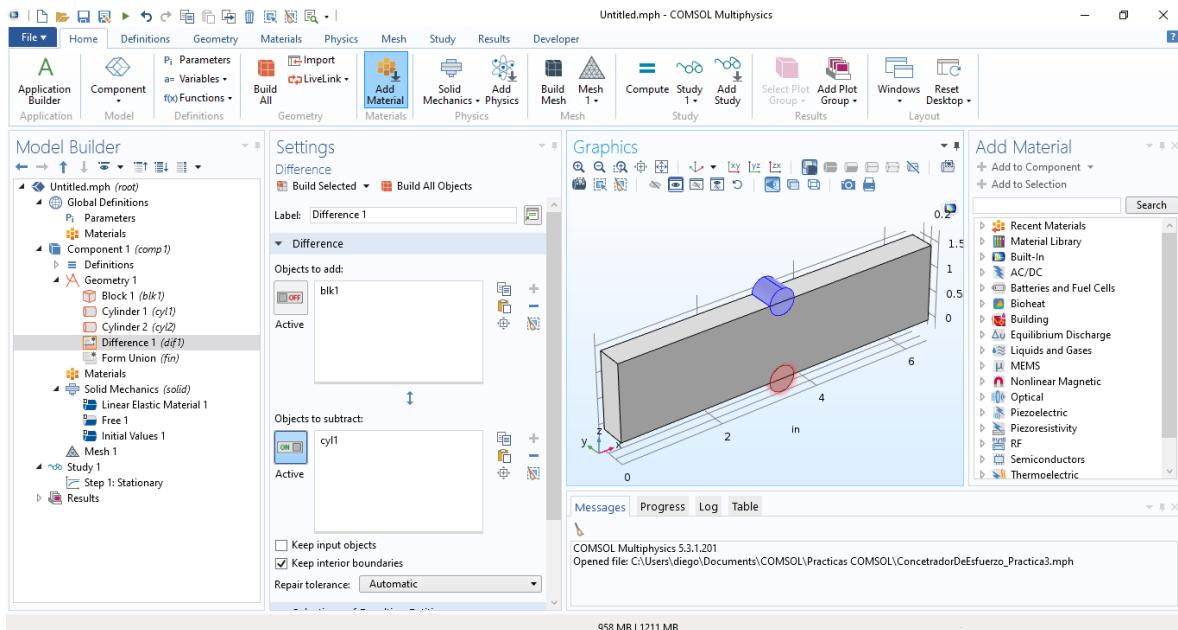
Primero selecciono el área que quedara.

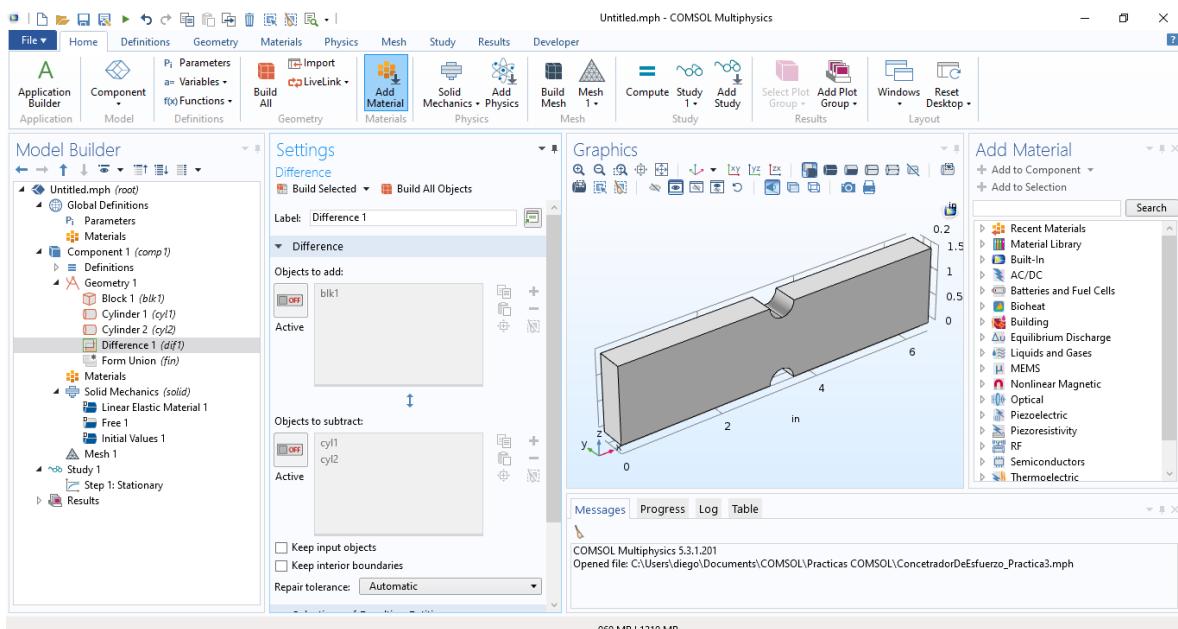
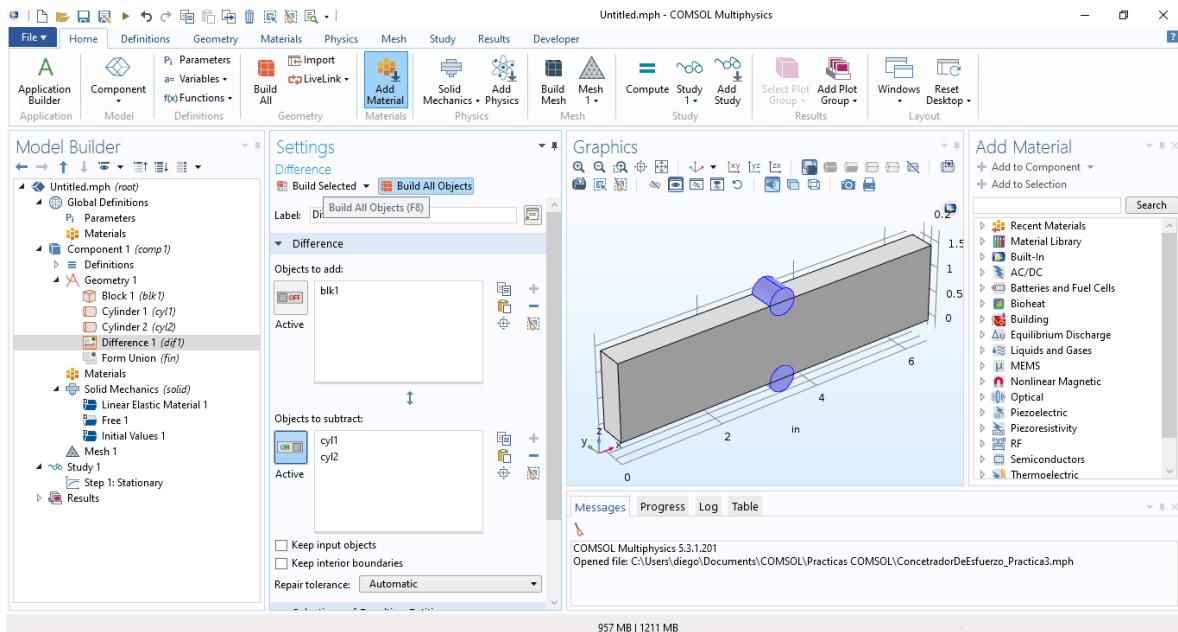


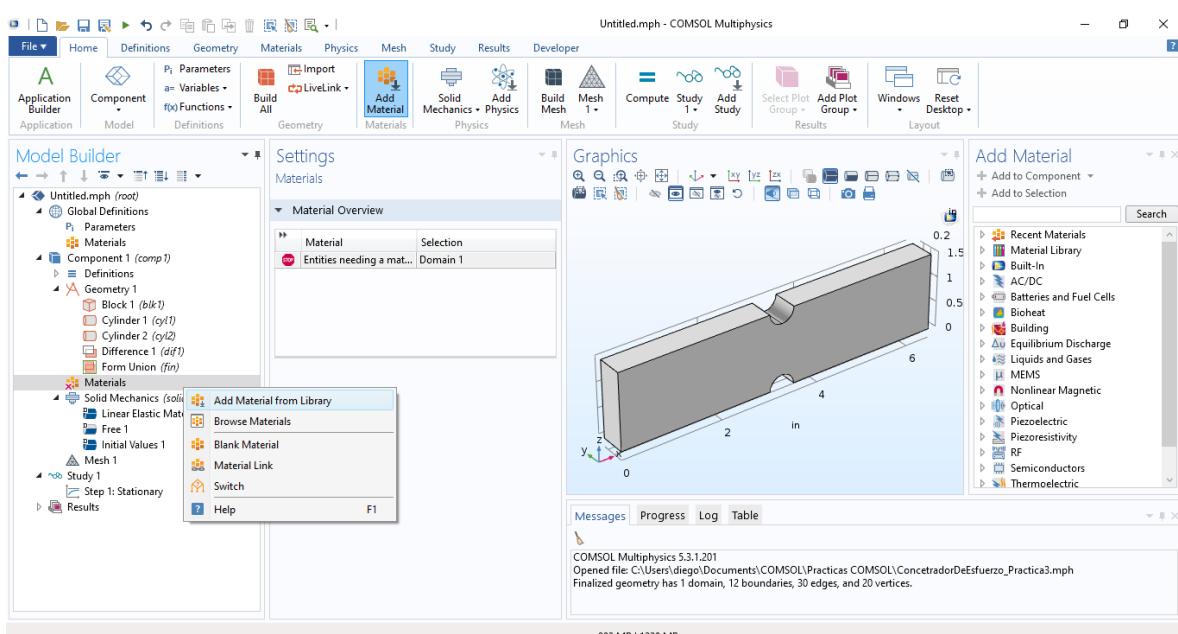
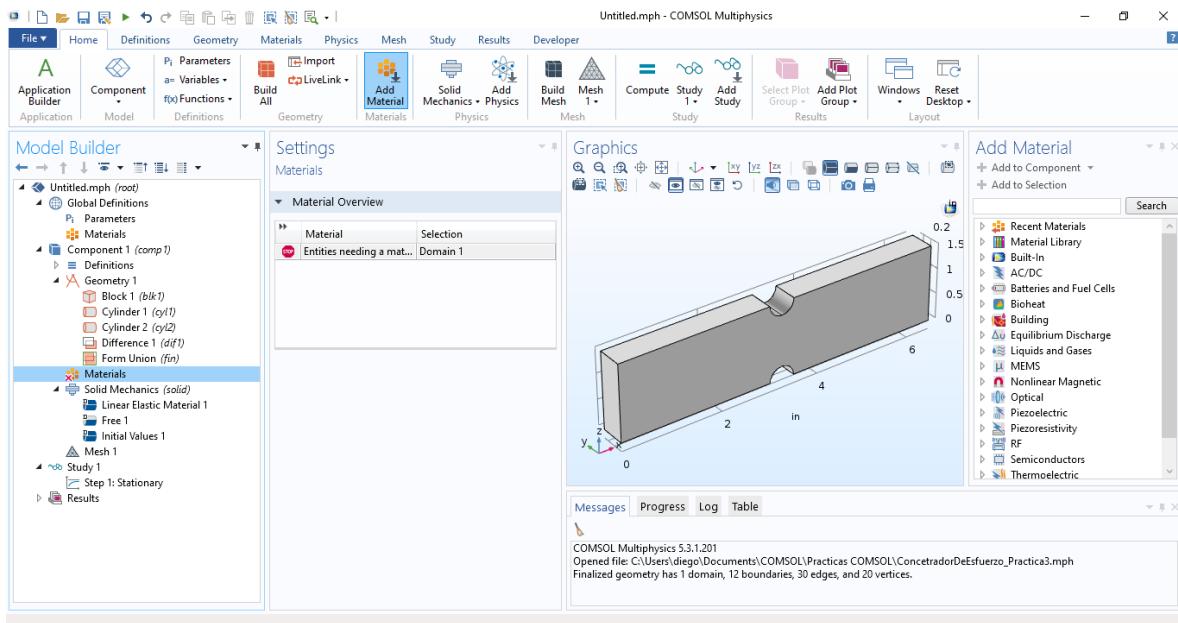


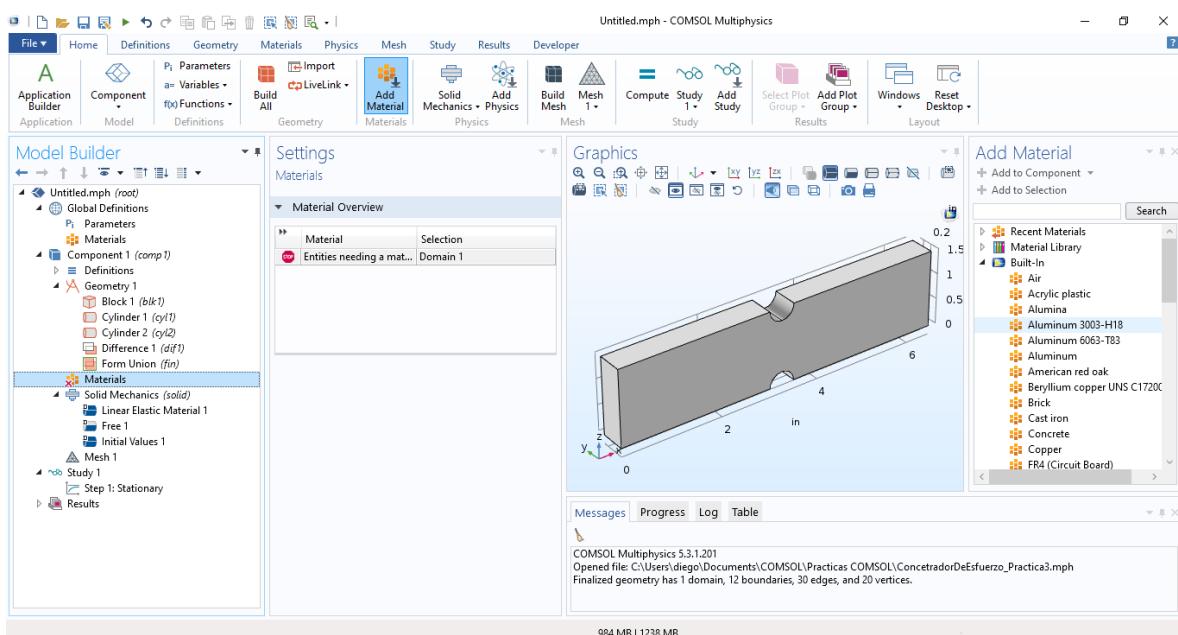
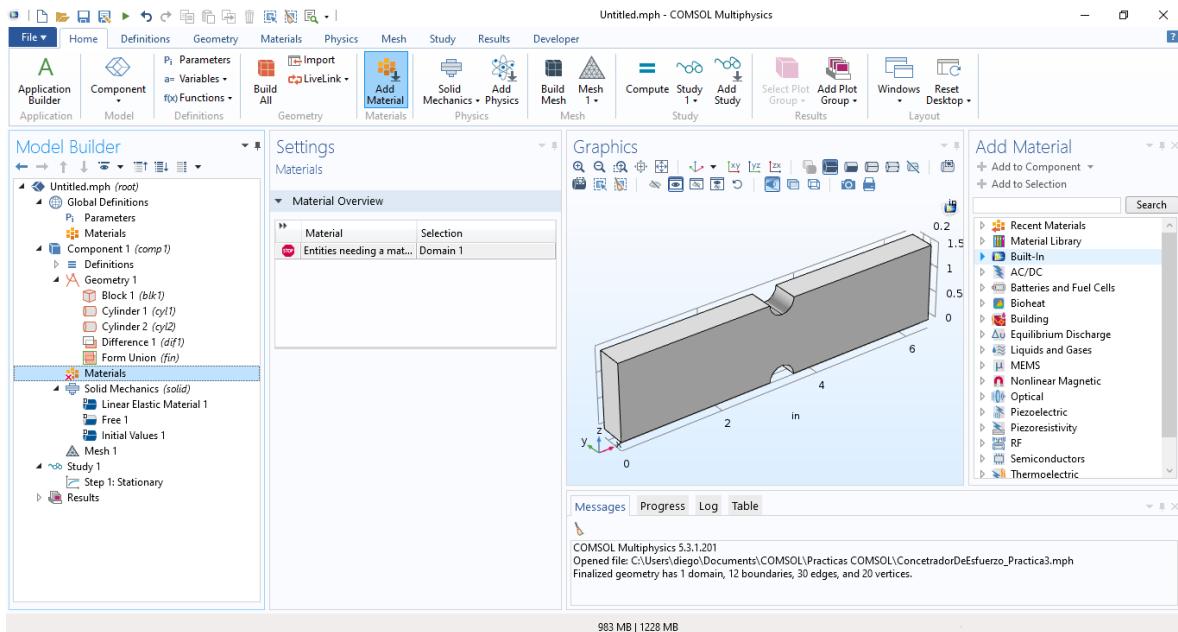
Y ahora seleccionamos lo que vayamos a quitar.

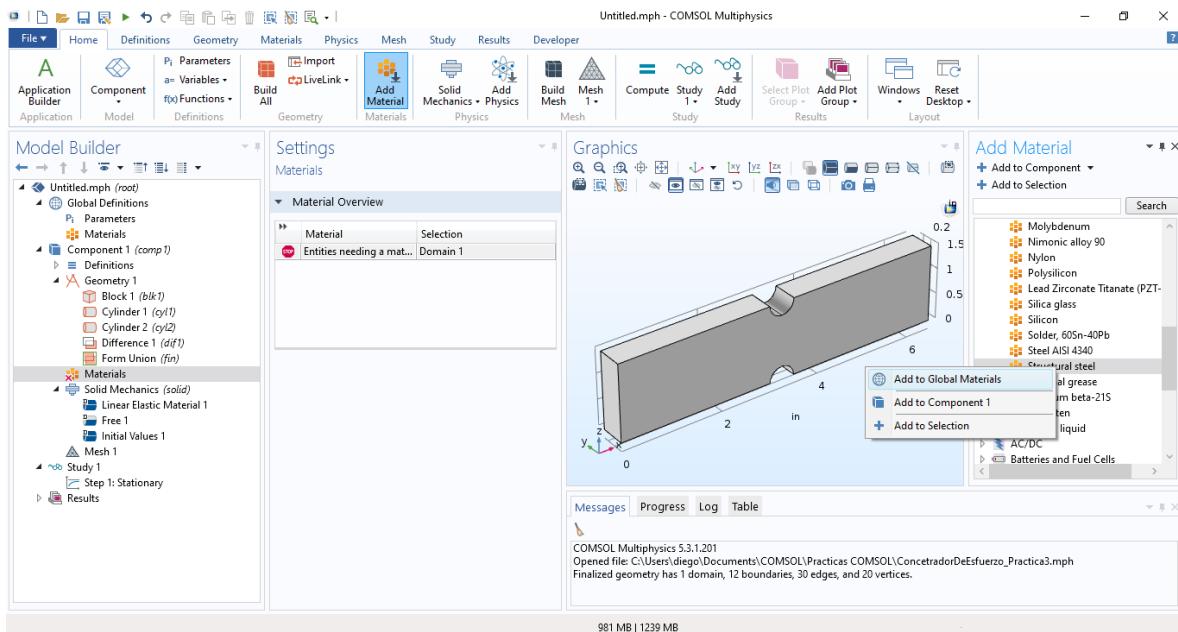




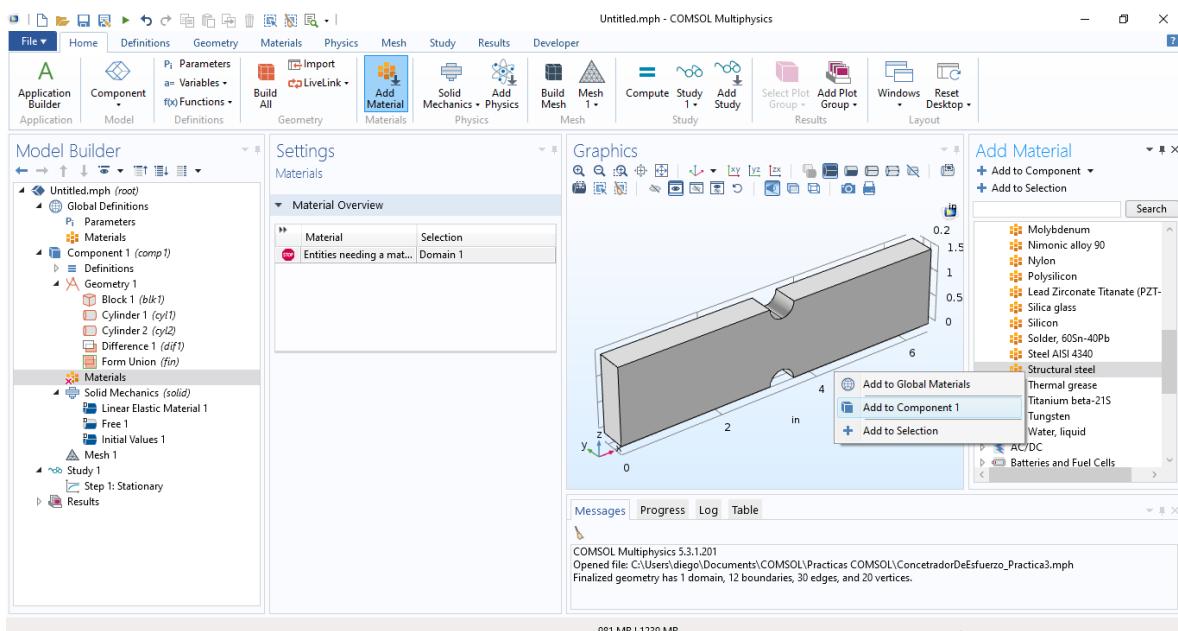




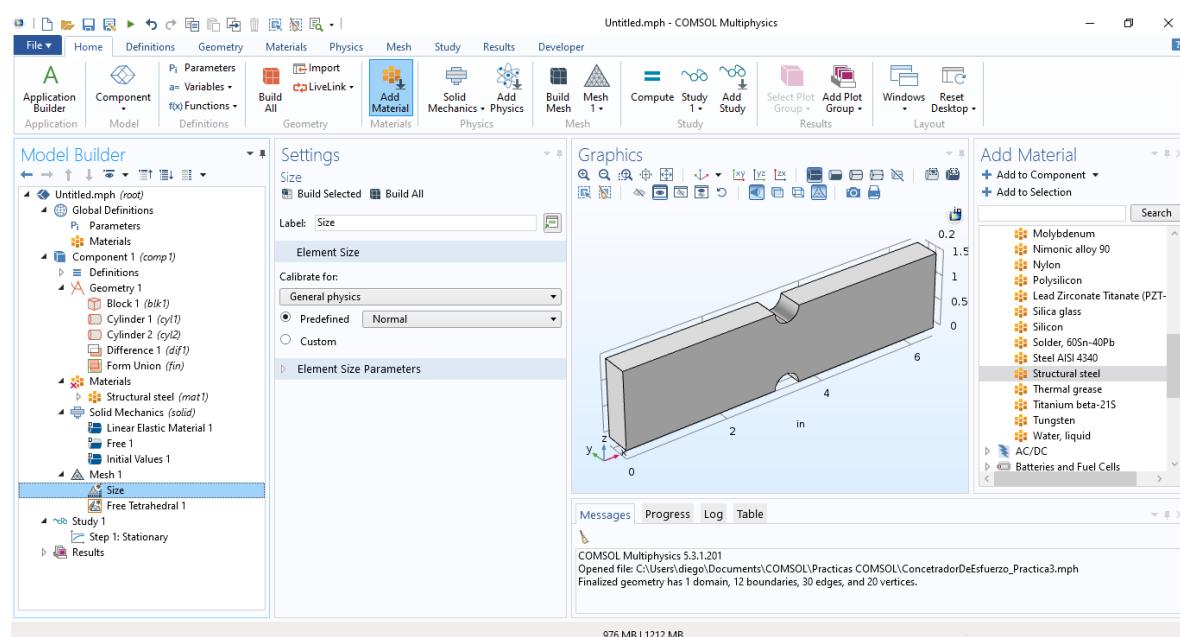
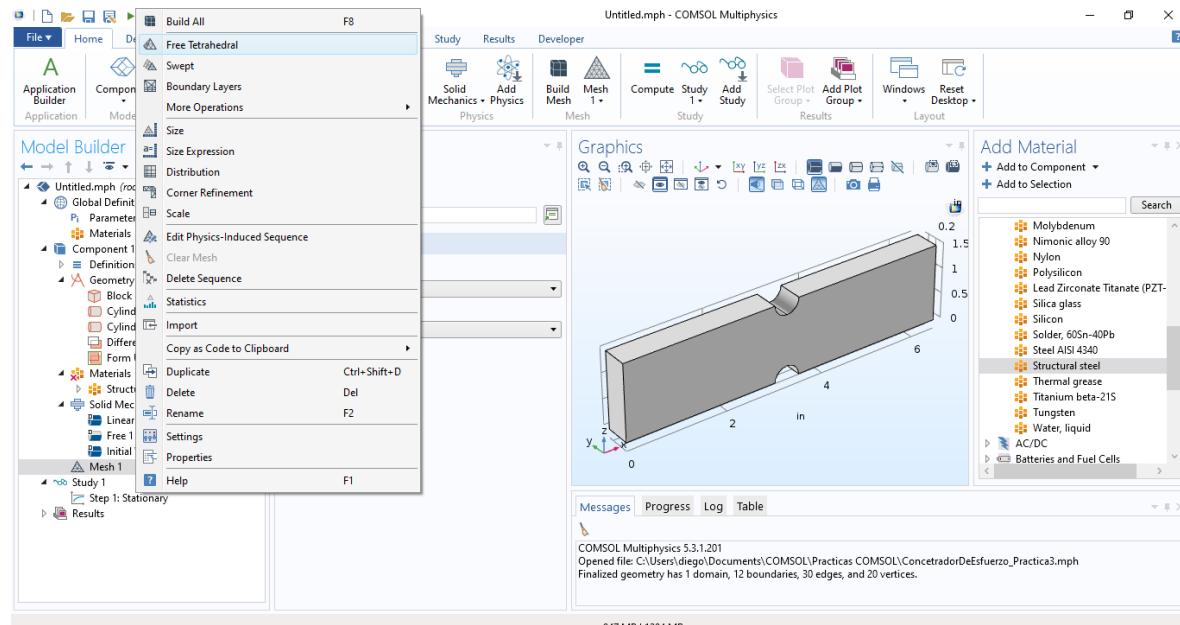


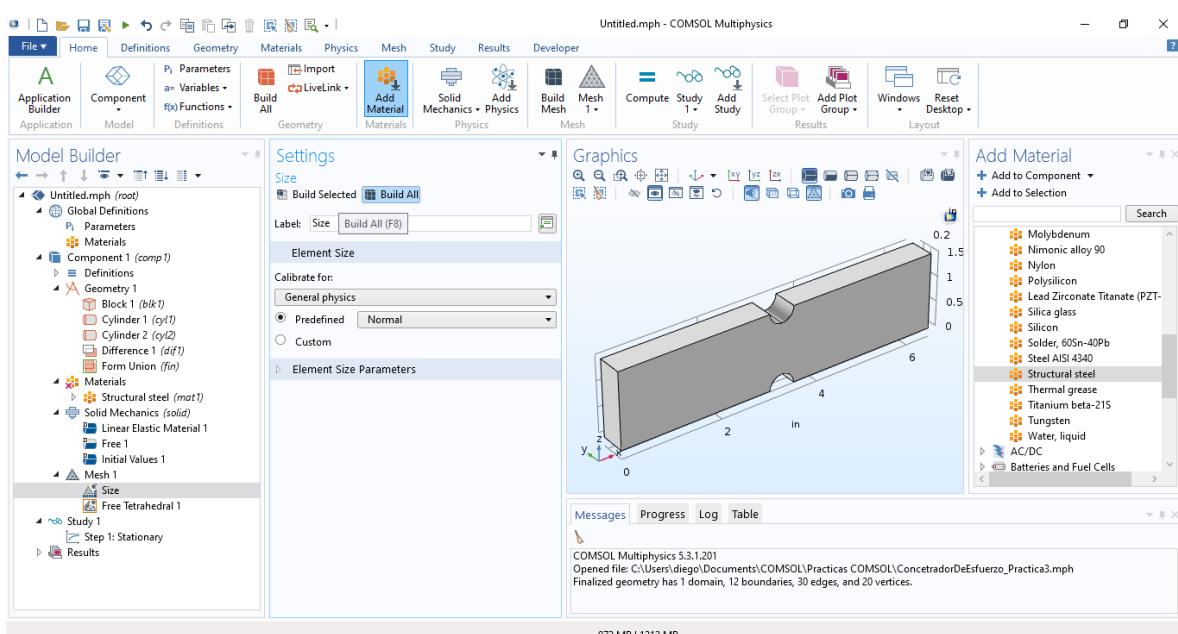
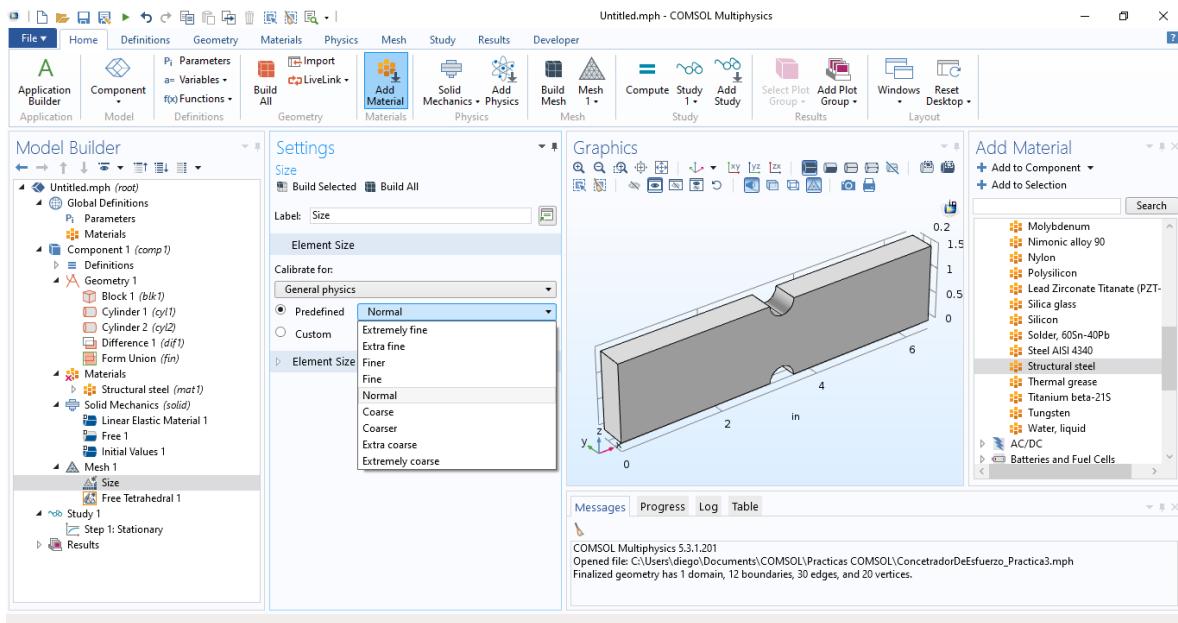


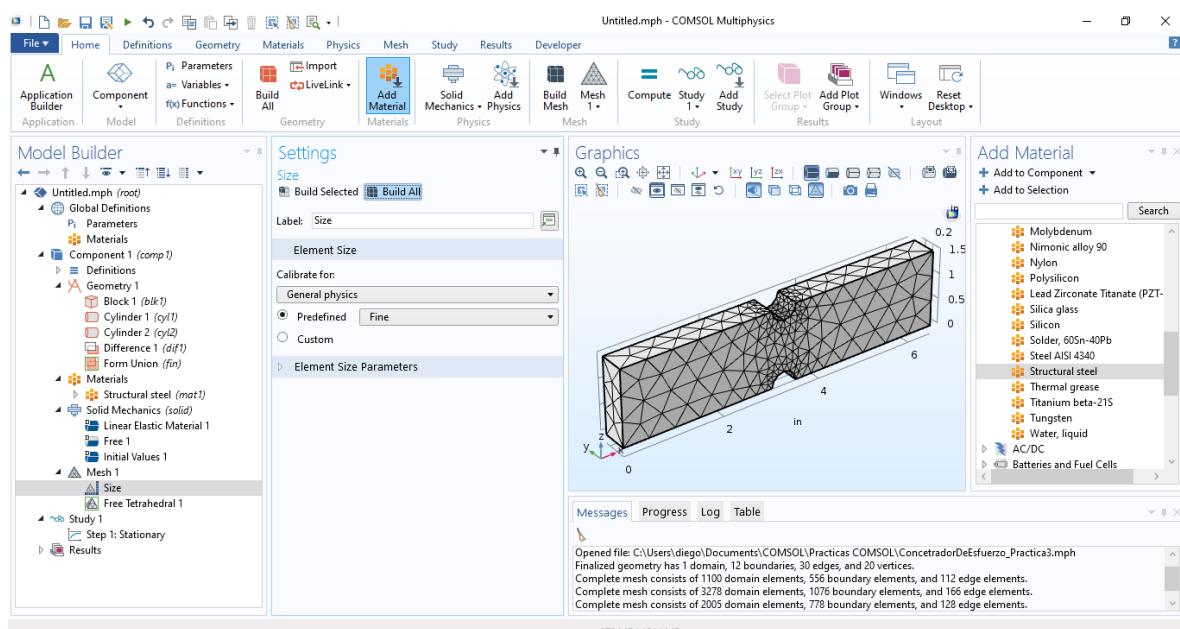
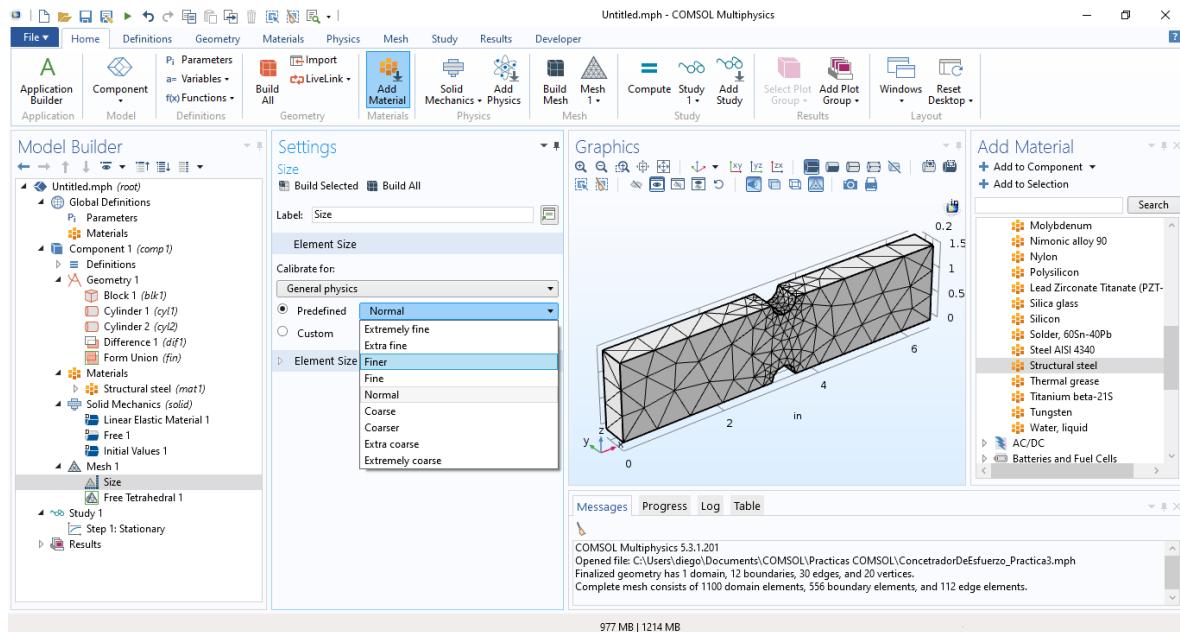
- **Add to selection:** para añadir materiales diferentes a cada parte.
- **Add to component:** para agregarlo a toda la estructura.

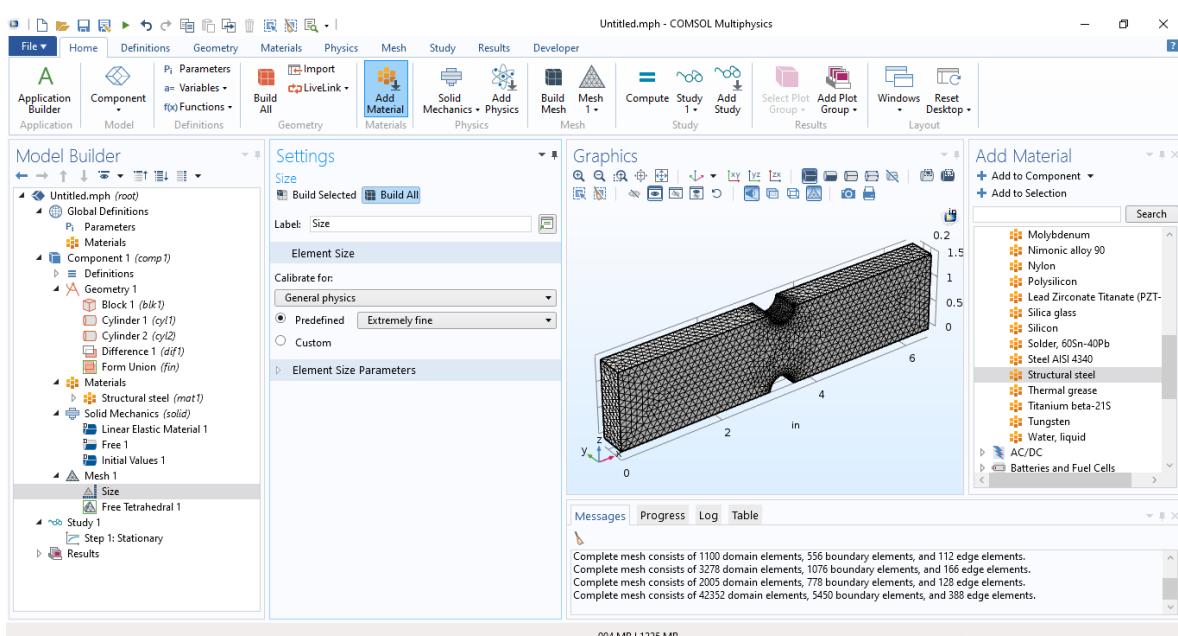
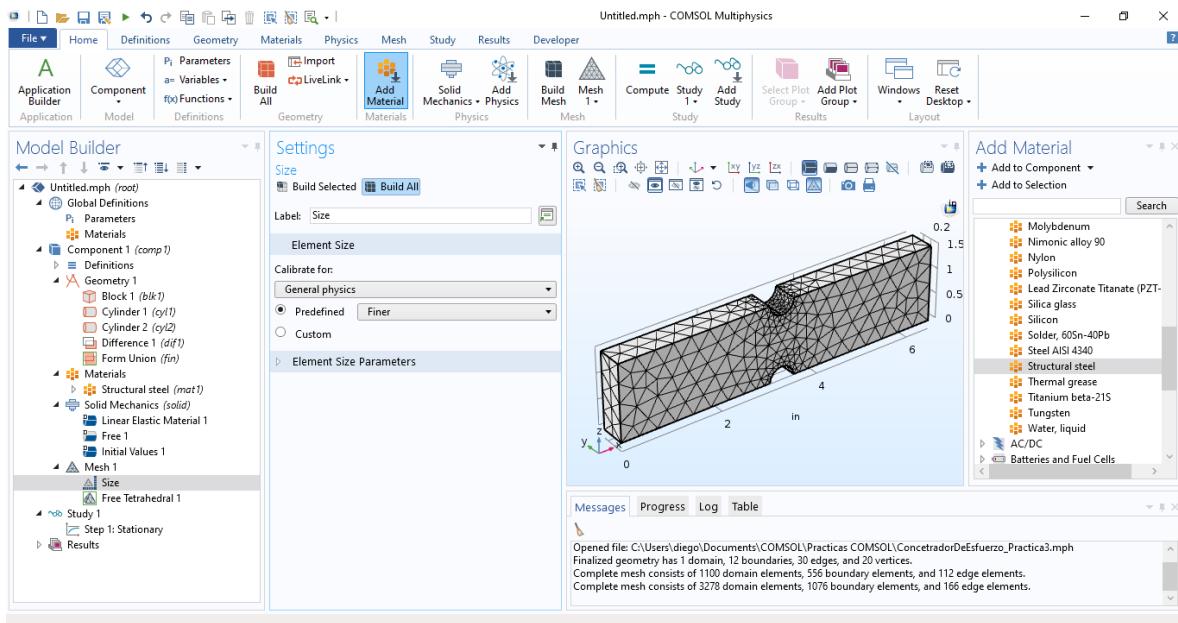


# ANÁLISIS MECÁNICO EN COMSOL:

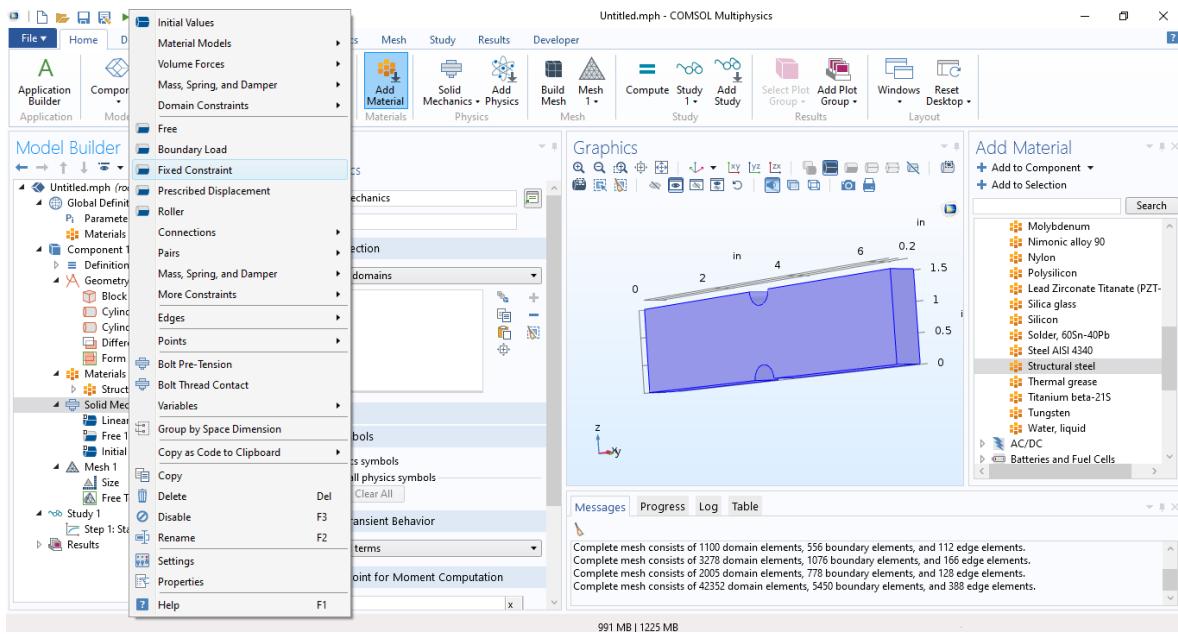




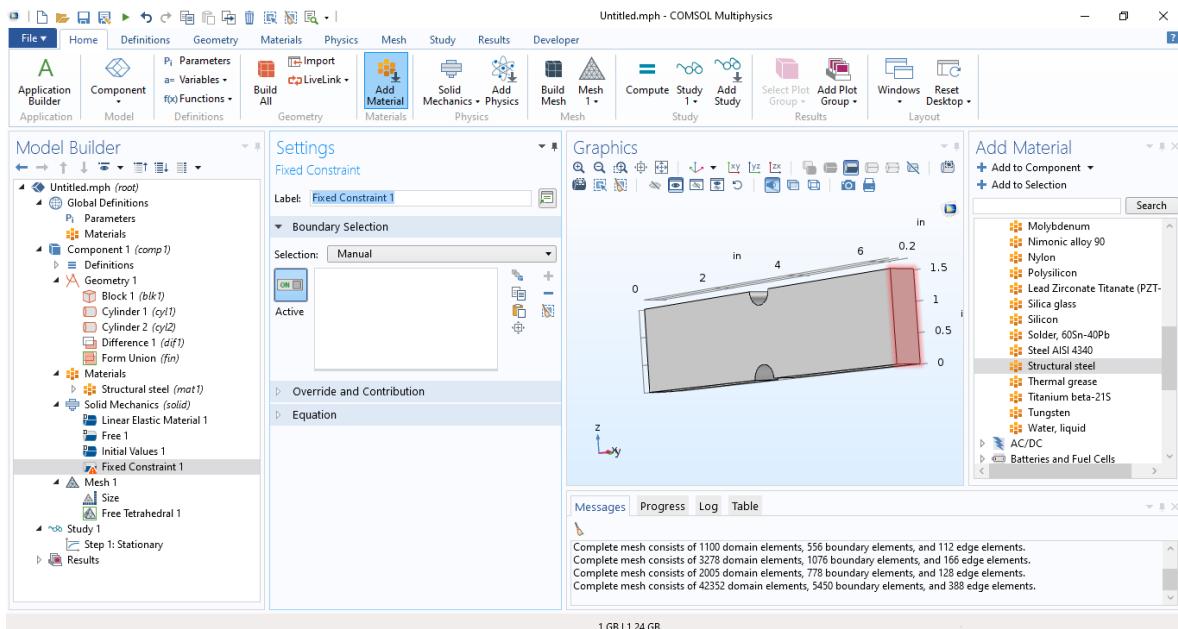


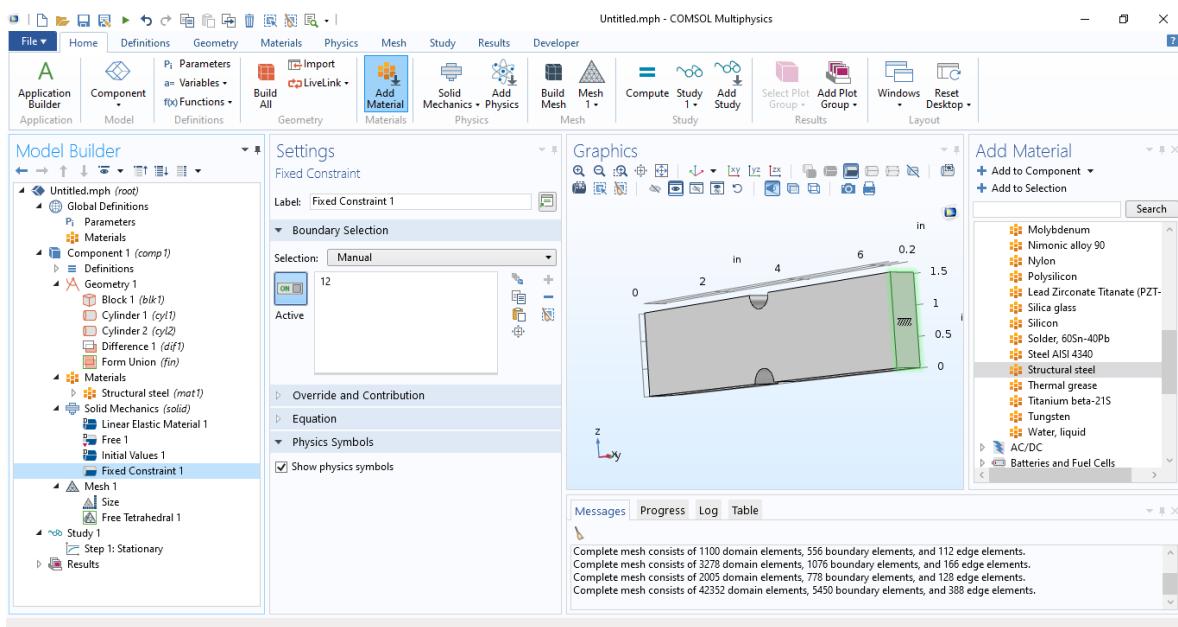
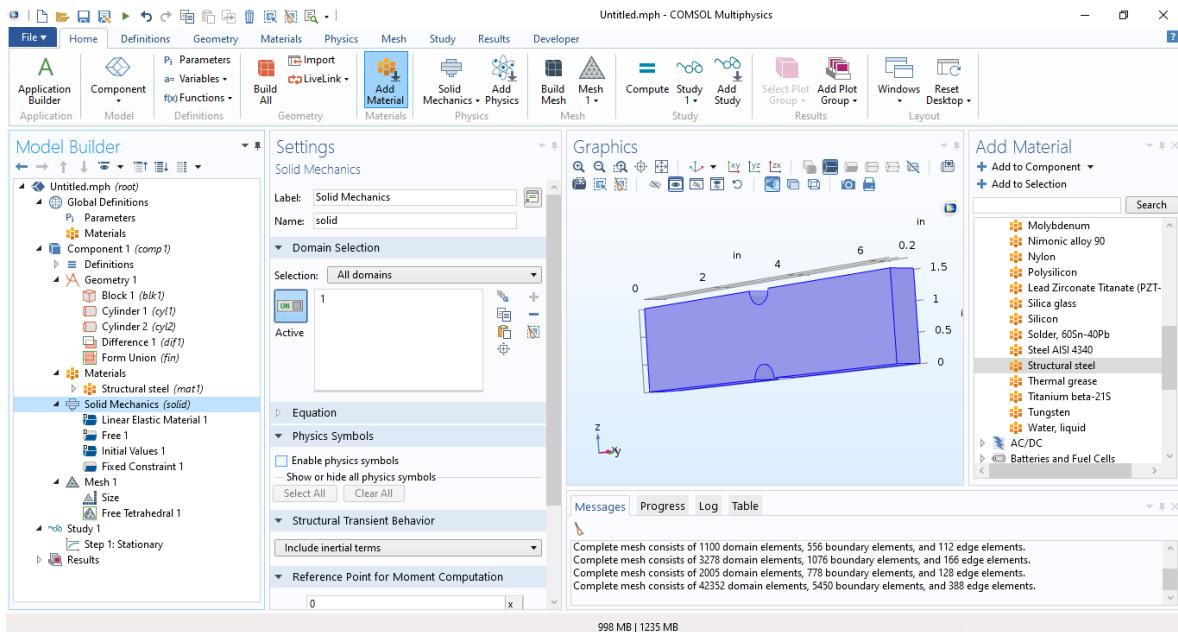


- La malla es para que nos dé un cálculo más exacto, pero se tarda más en hacer el cálculo.
- Cuando la carga es lineal (constante) no hay problema, pero si es no lineal (varía la carga respecto al tiempo) se debería hacer más cosas.

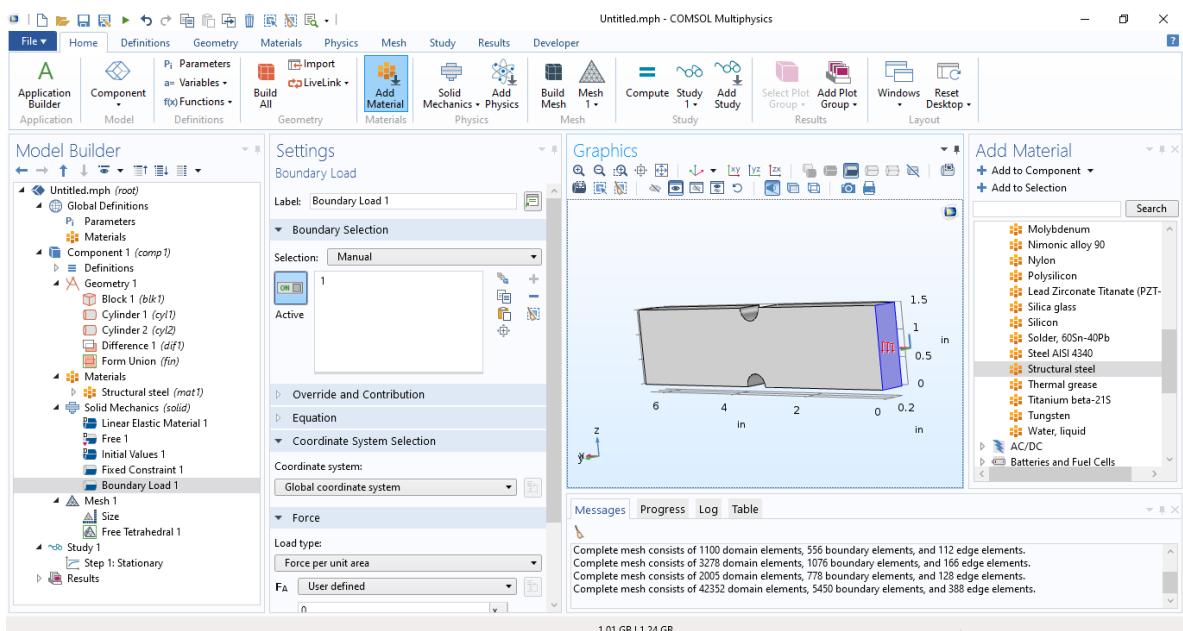
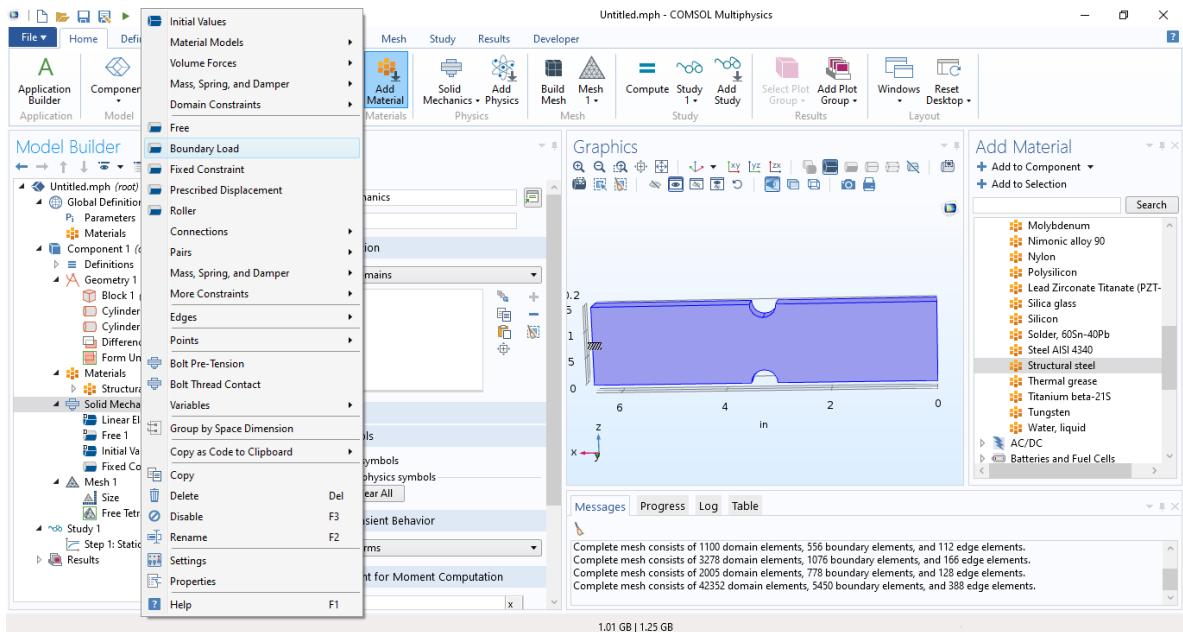


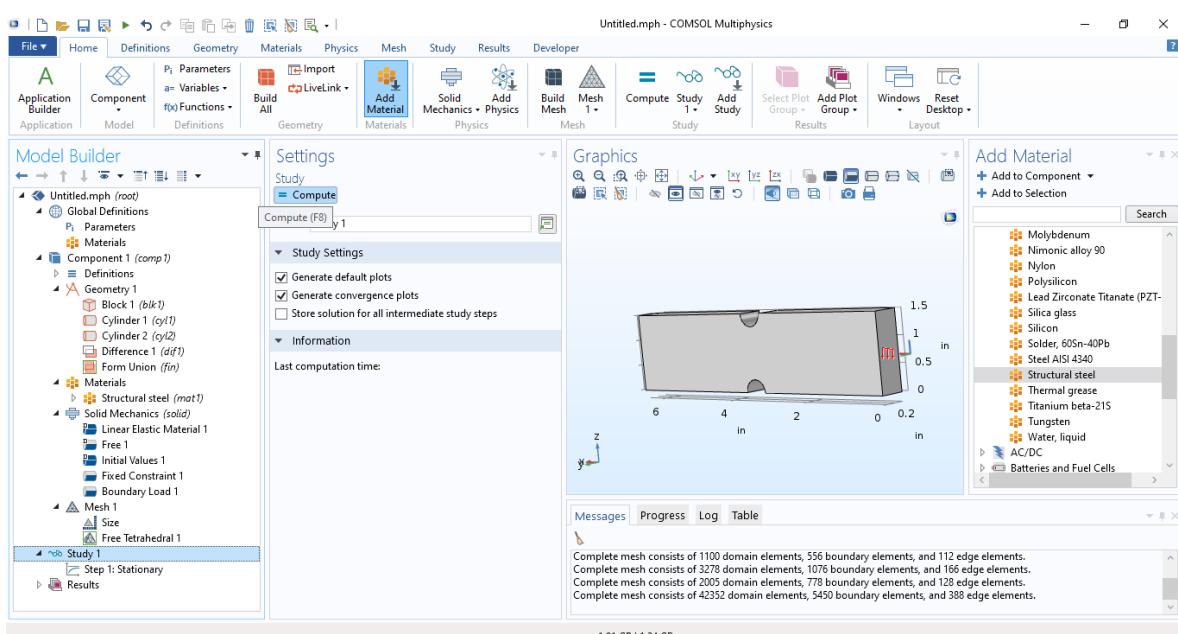
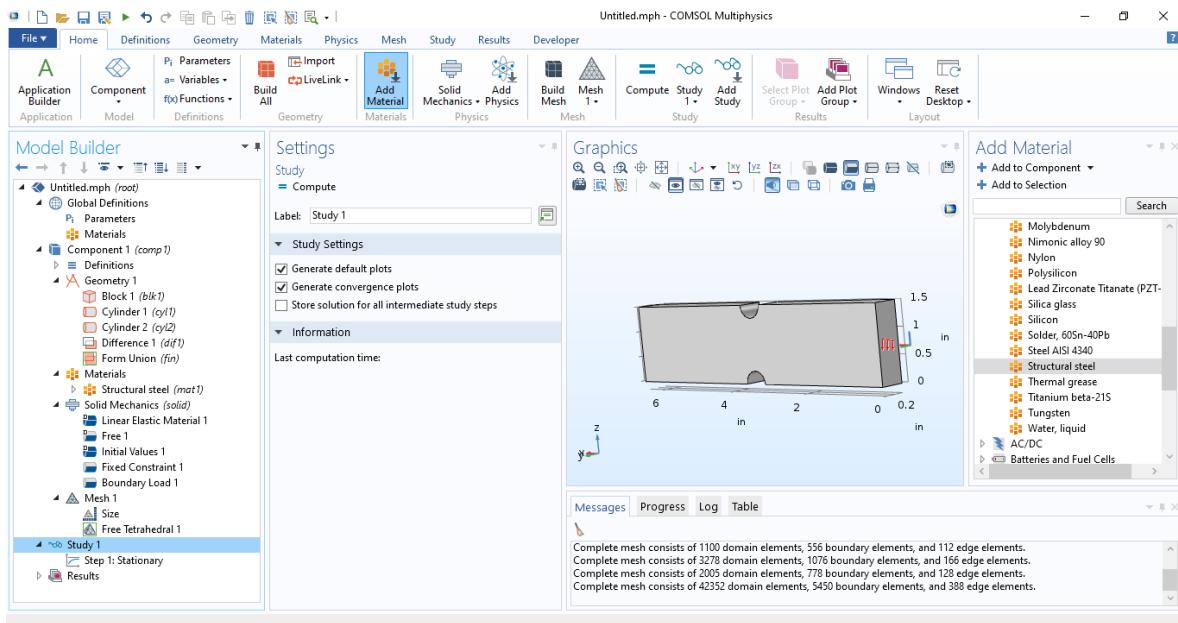
Ahora indico el área que estará fija durante la aplicación de fuerza, osea el apoyo.

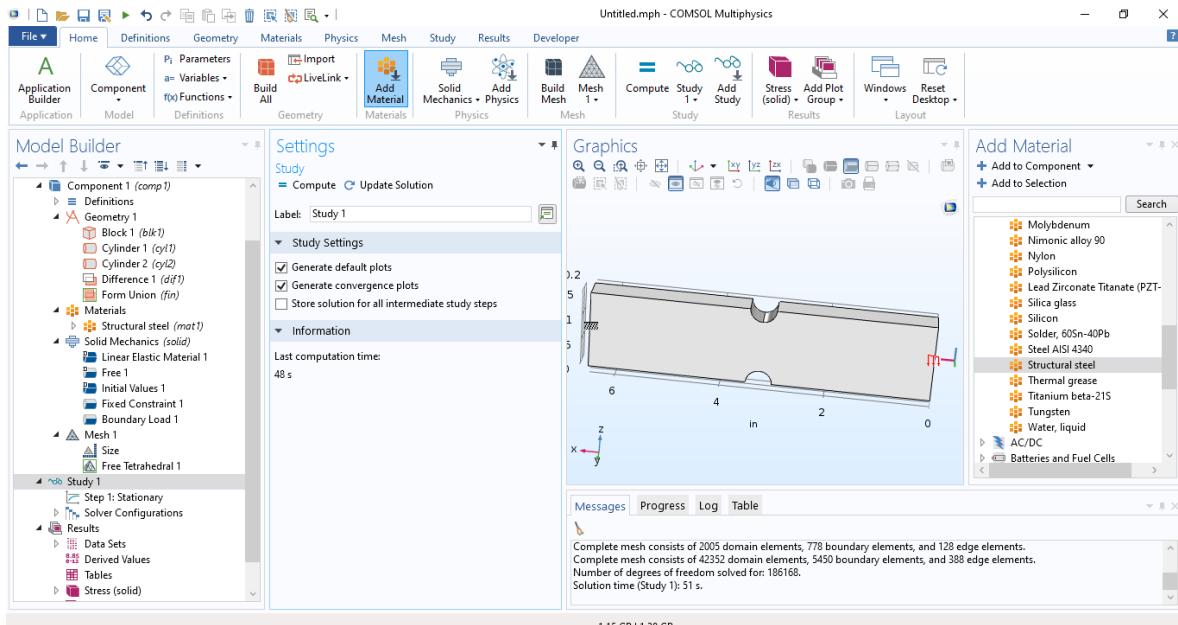
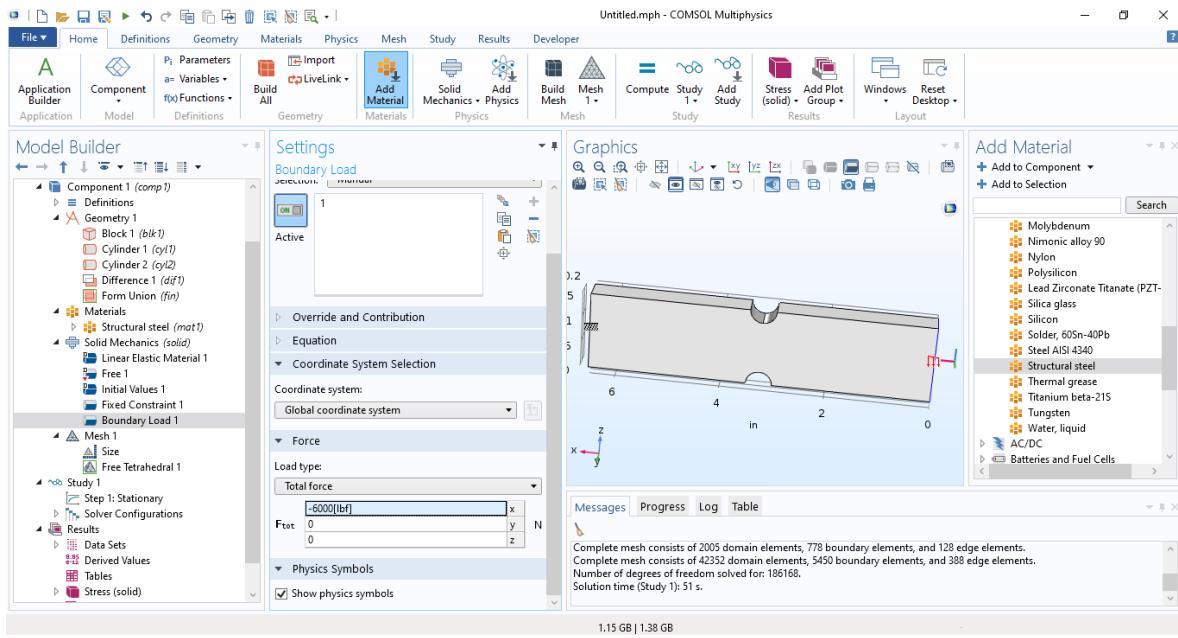




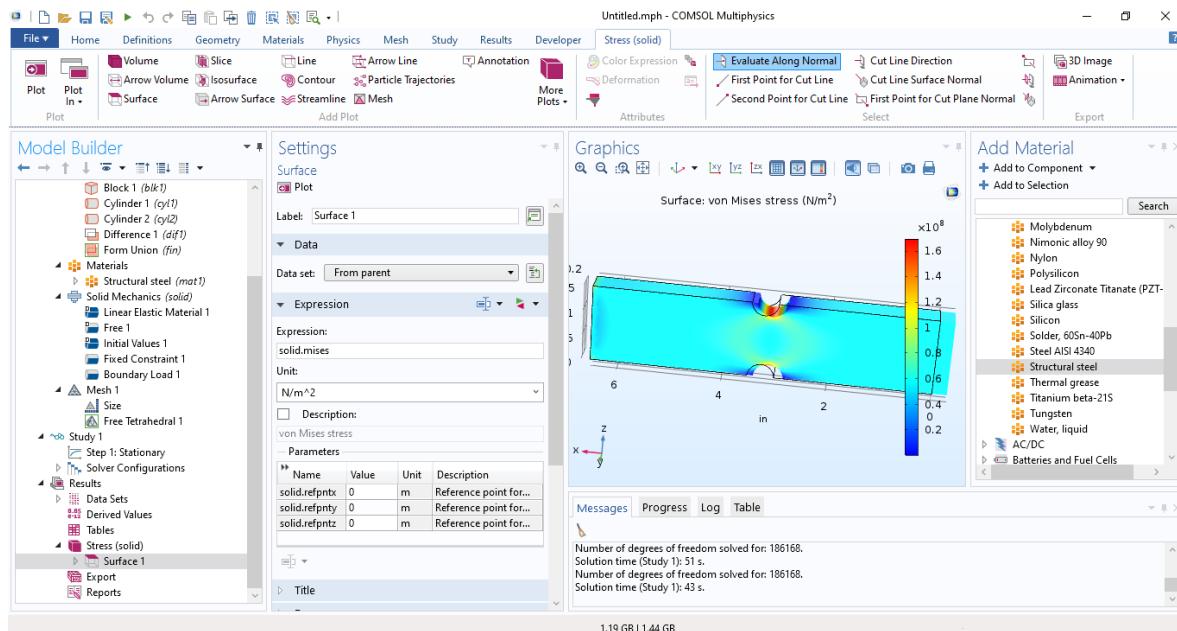
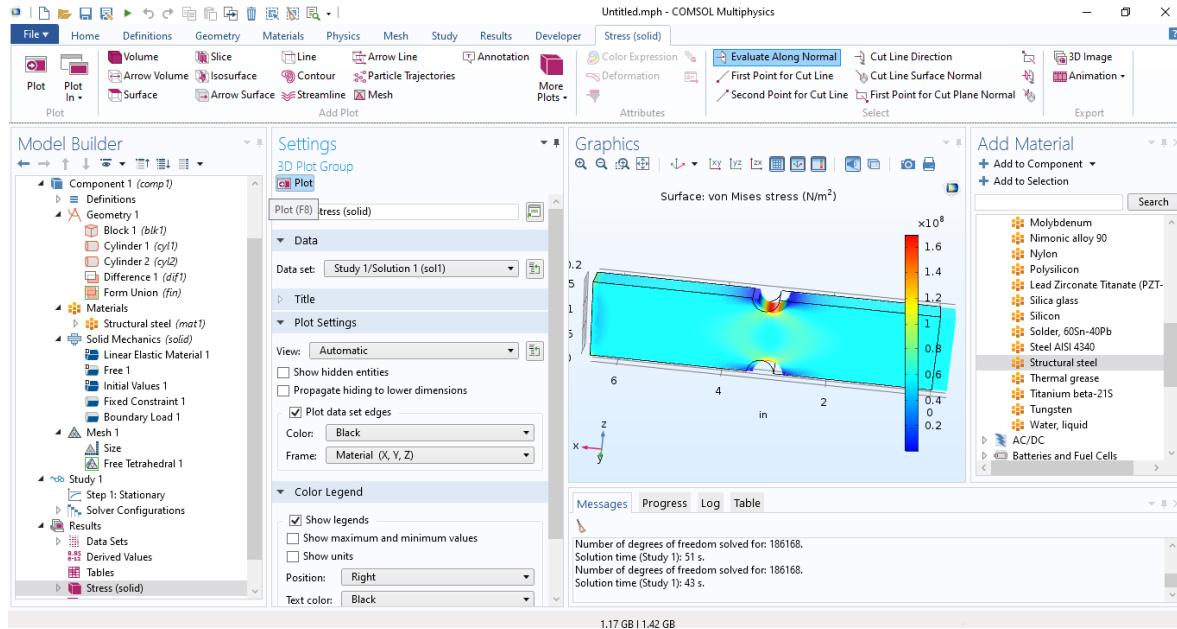
Ya habiendo indicado la posición del apoyo o empotramiento se coloca por medio de la opción de Boundary Load la carga, osea la fuerza aplicada.

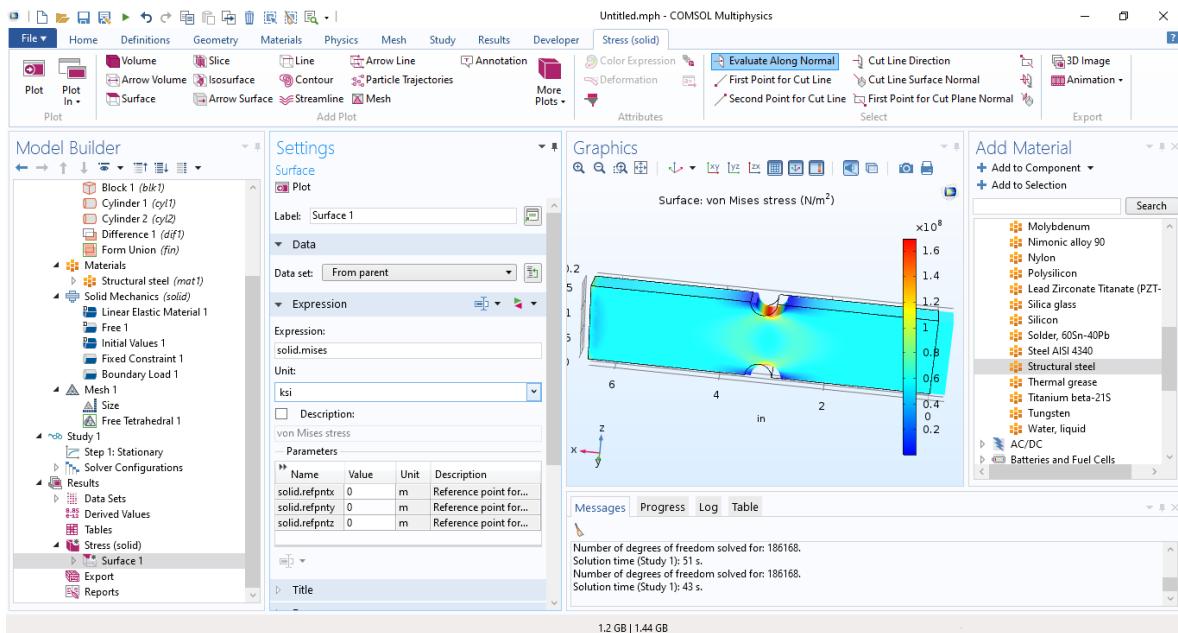
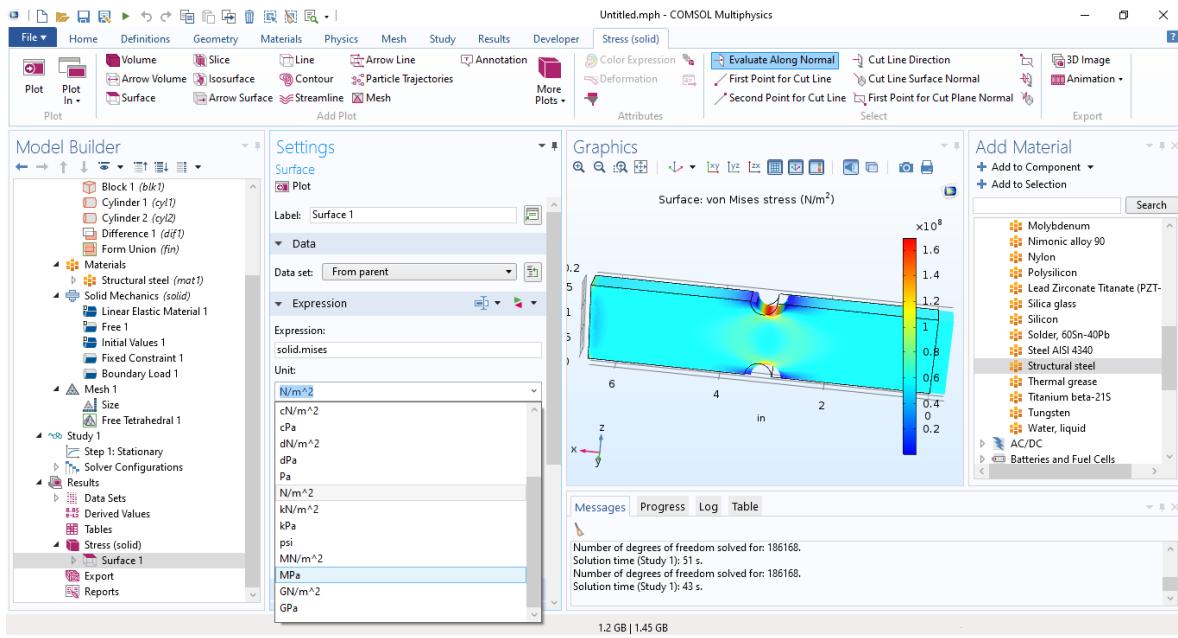


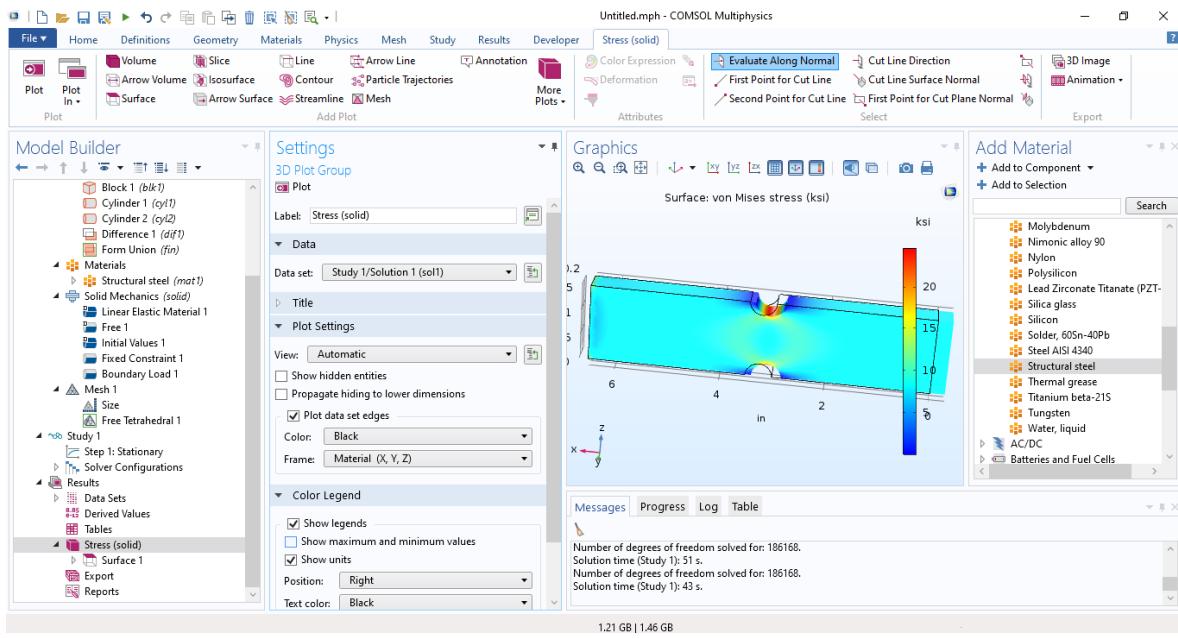




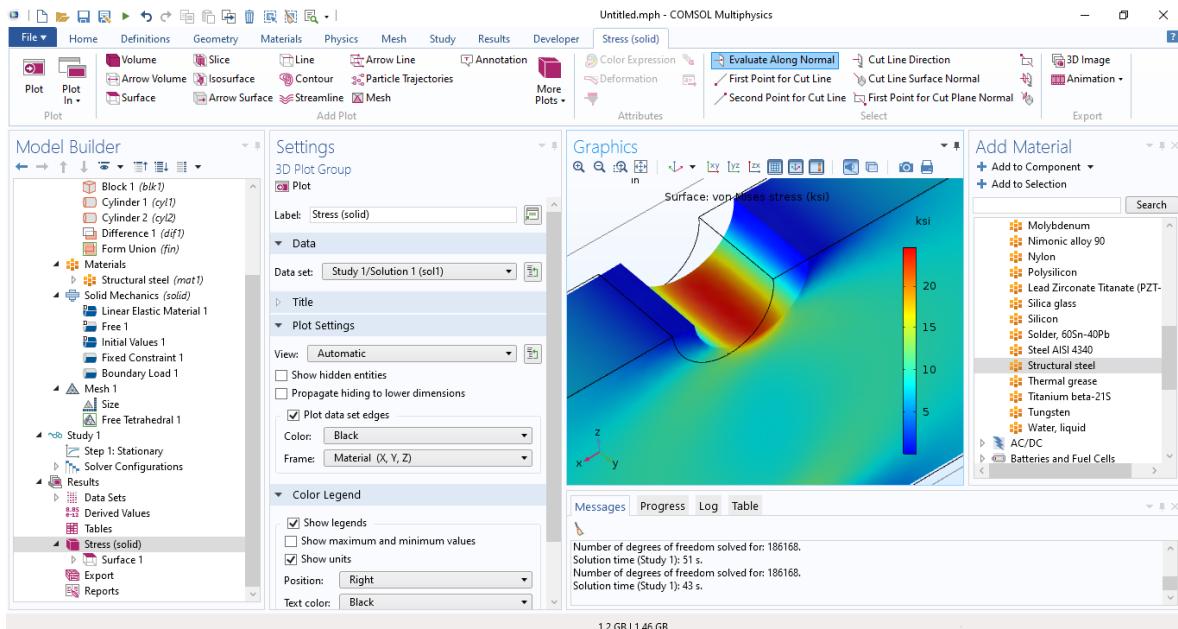
# RESULTADO DEL ELEMENTO FINITO EN COMSOL:

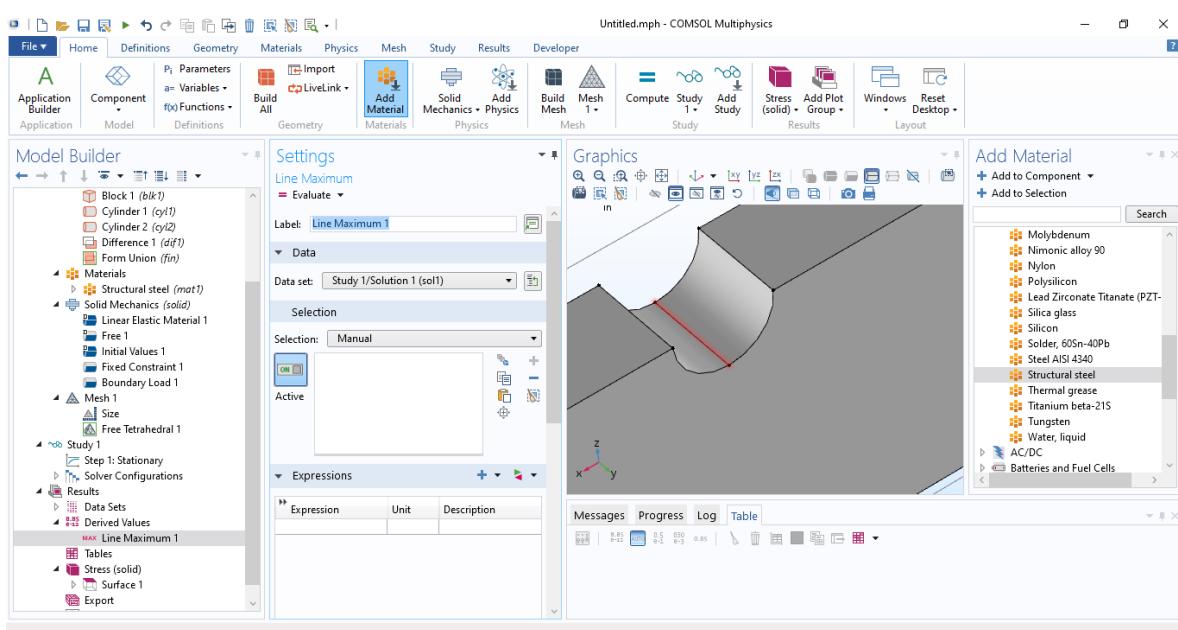
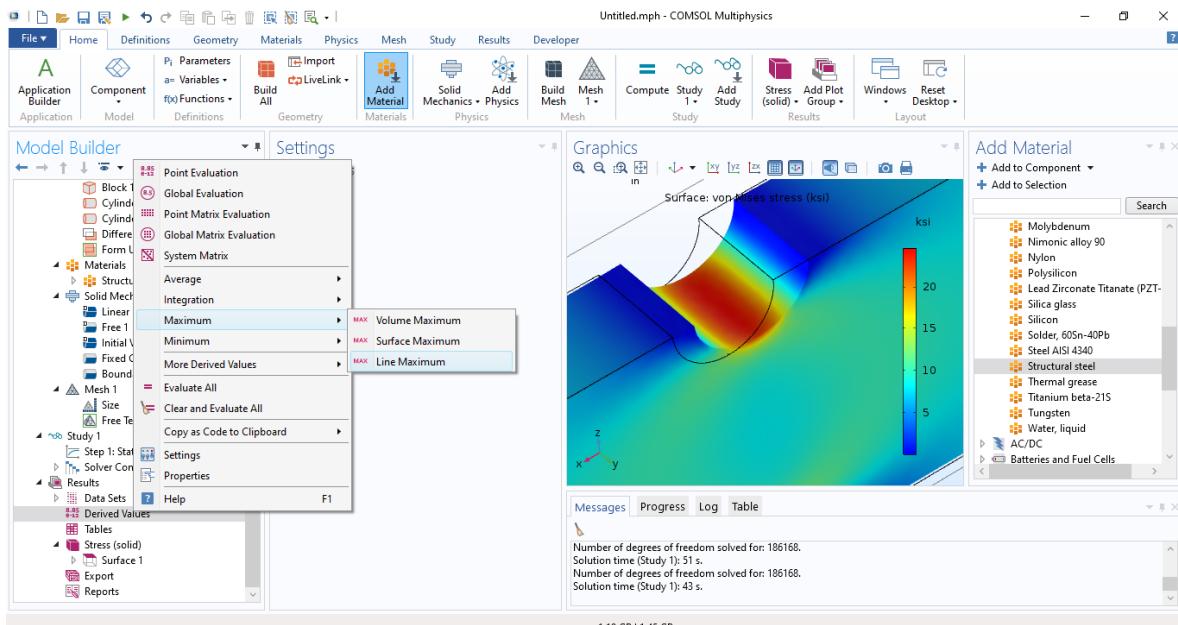


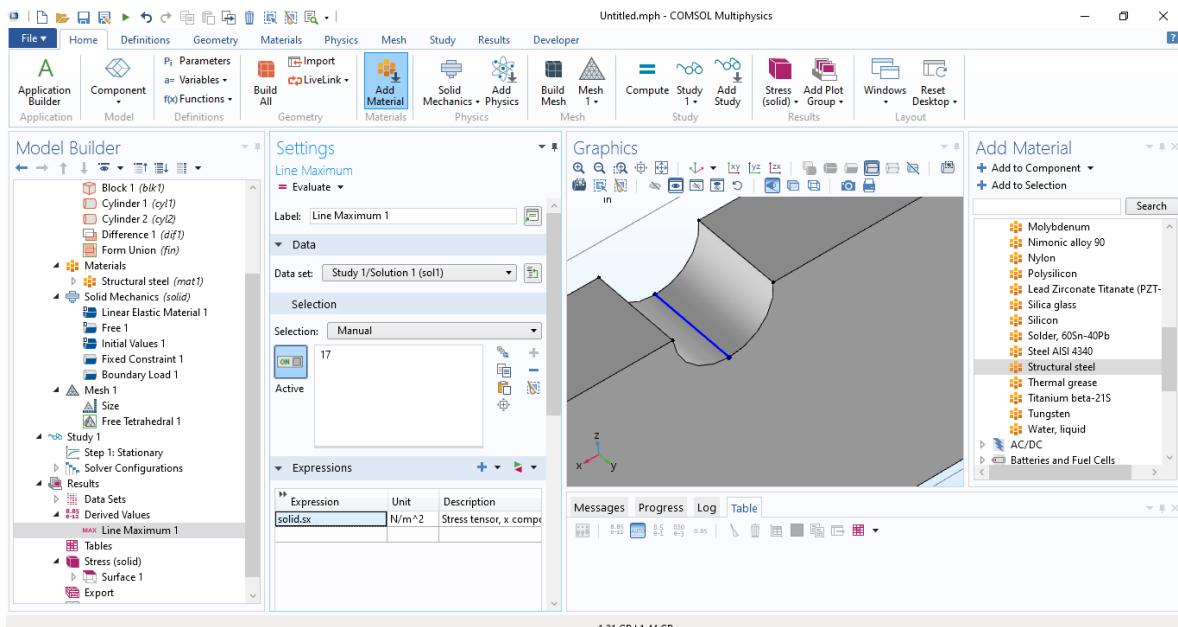
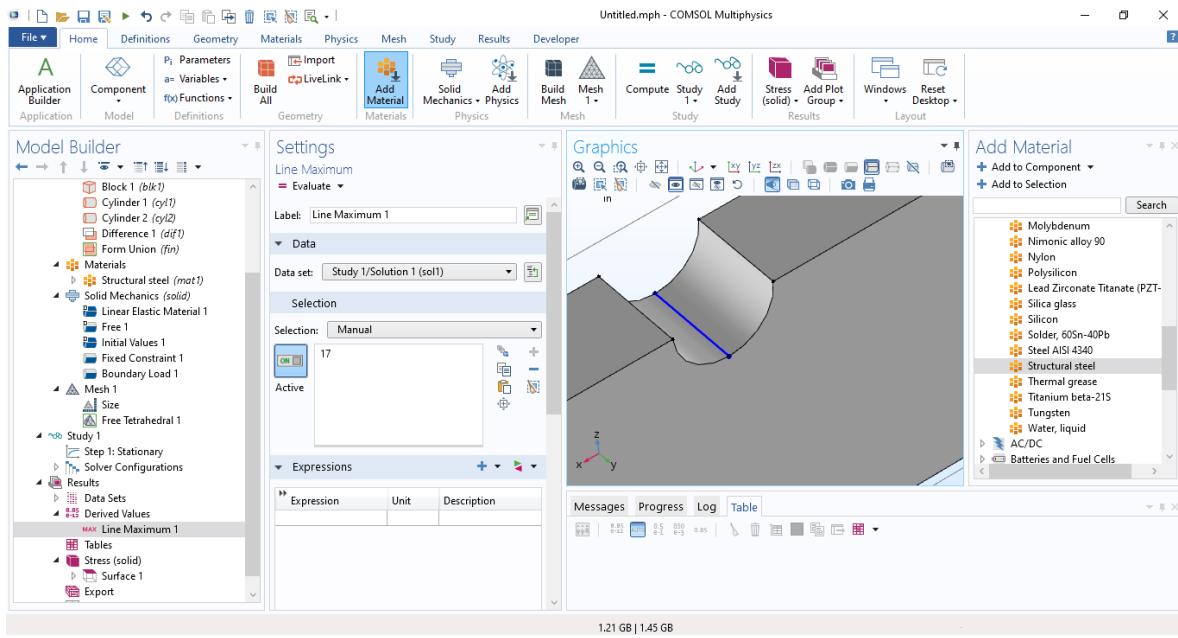


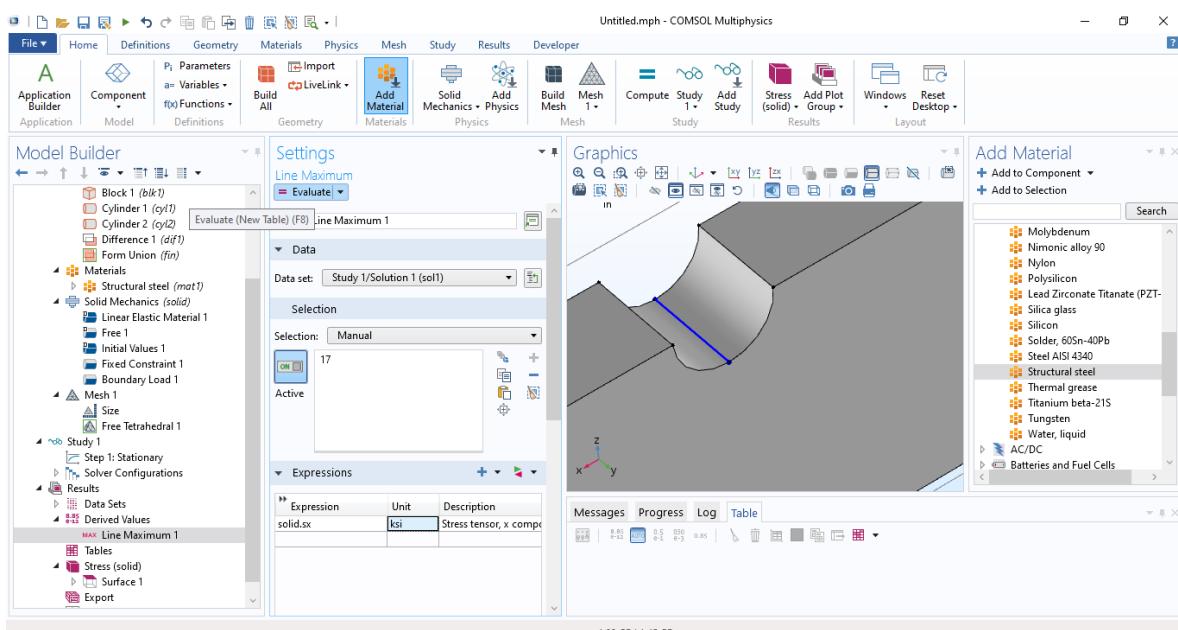
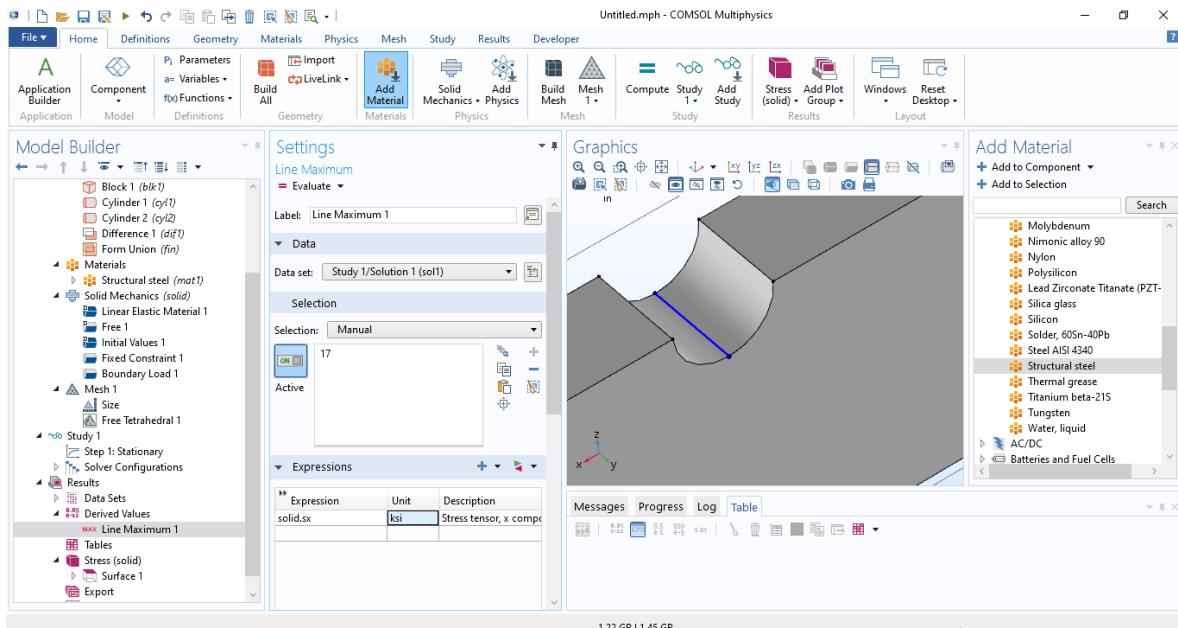


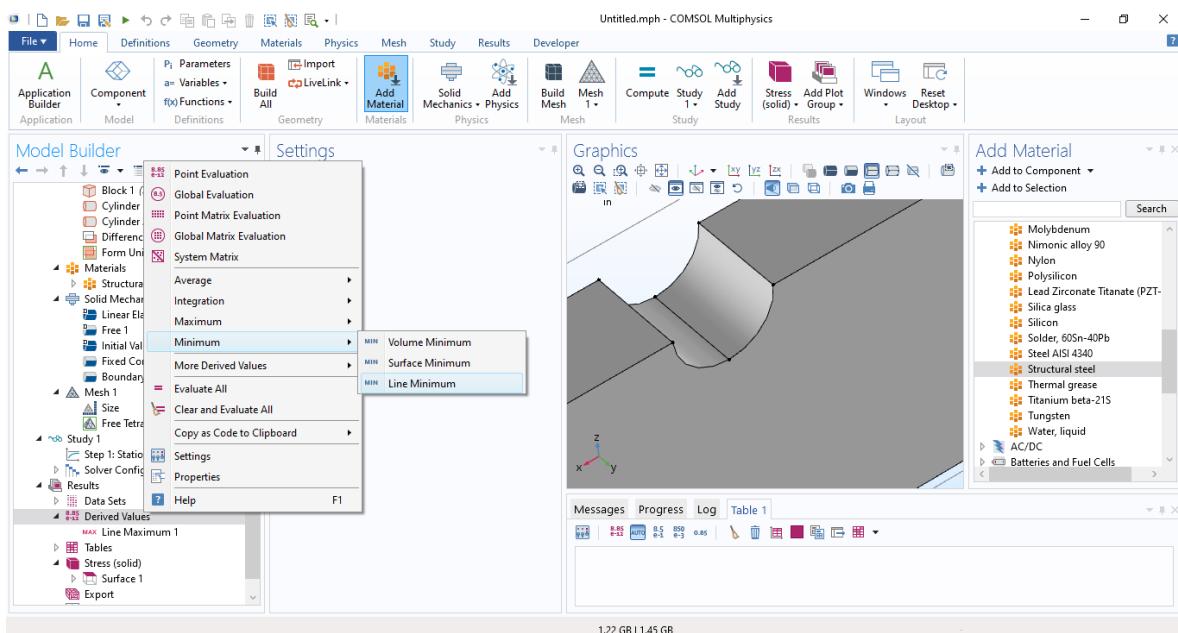
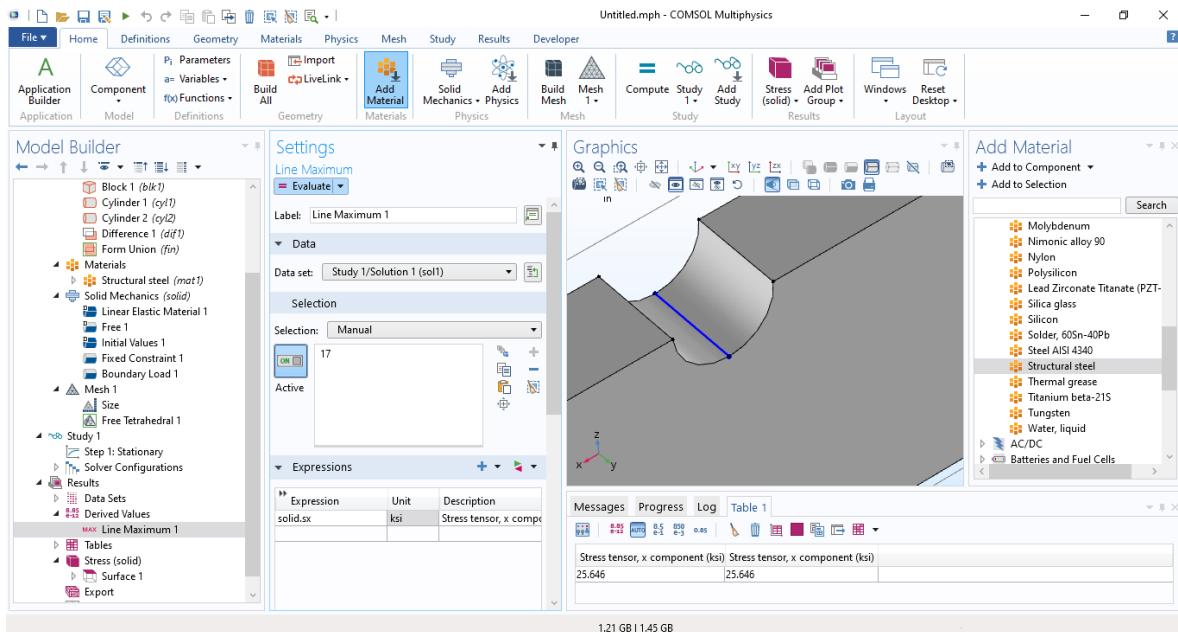
Se puede observar en el modelo como todo el esfuerzo se concentra en las partes del concentrador de esfuerzo, por ello es que se debe evitar crear esquinas o agujeros en nuestras piezas, solo se dejan cuando sea de suma importancia que estén ahí como por ejemplo para poner un tornillo, remache, etc.

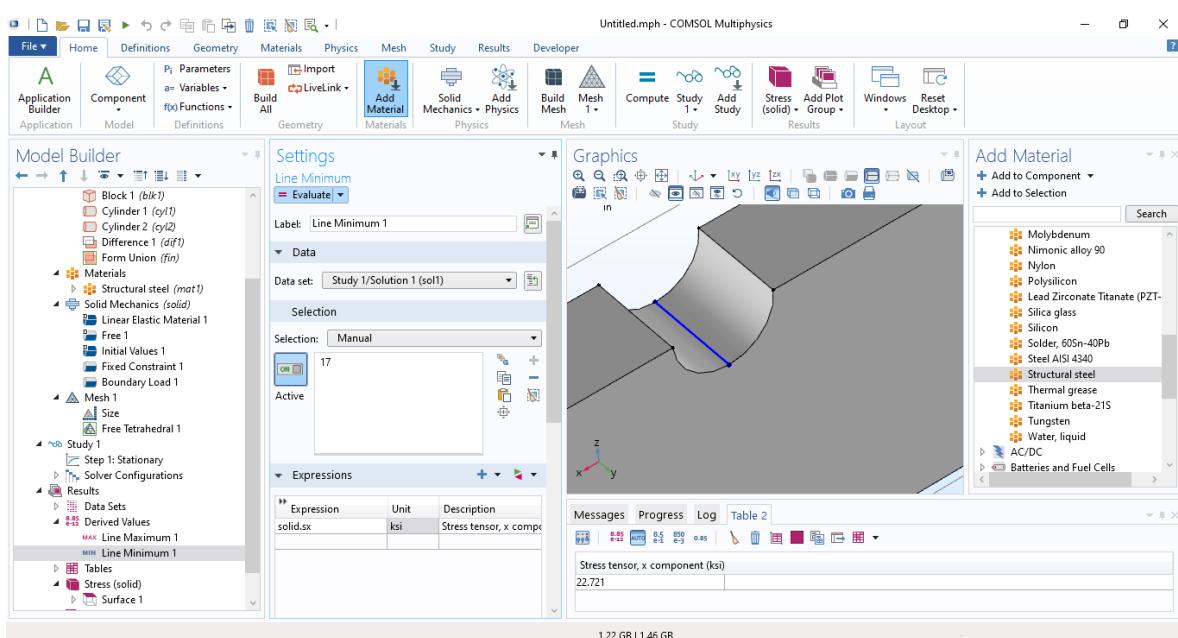
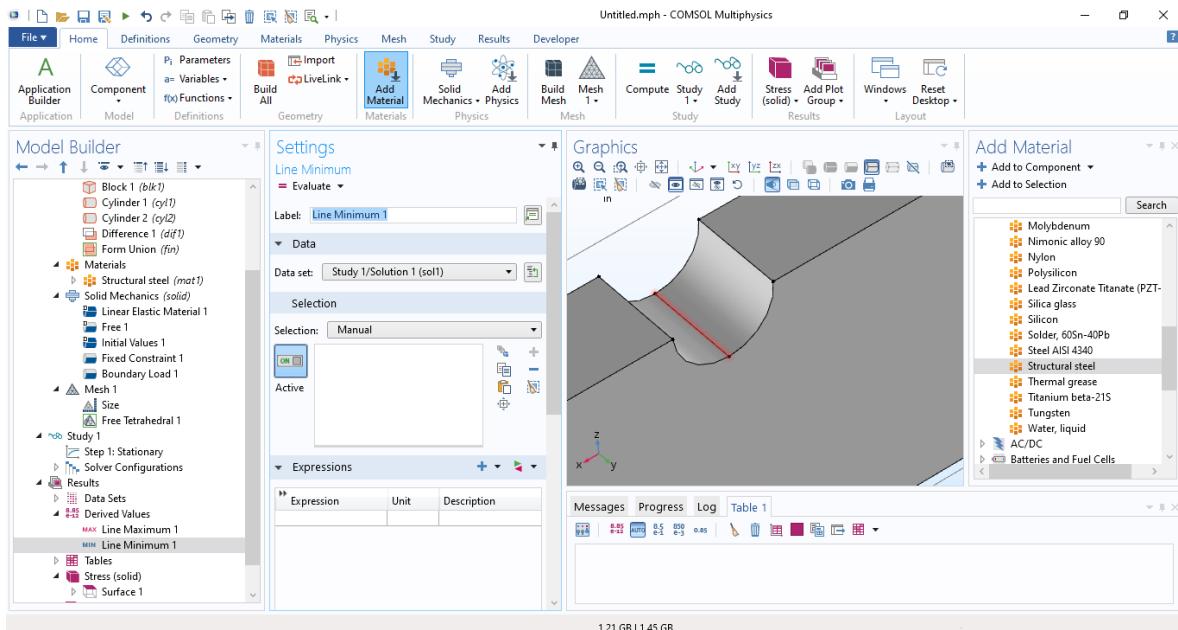


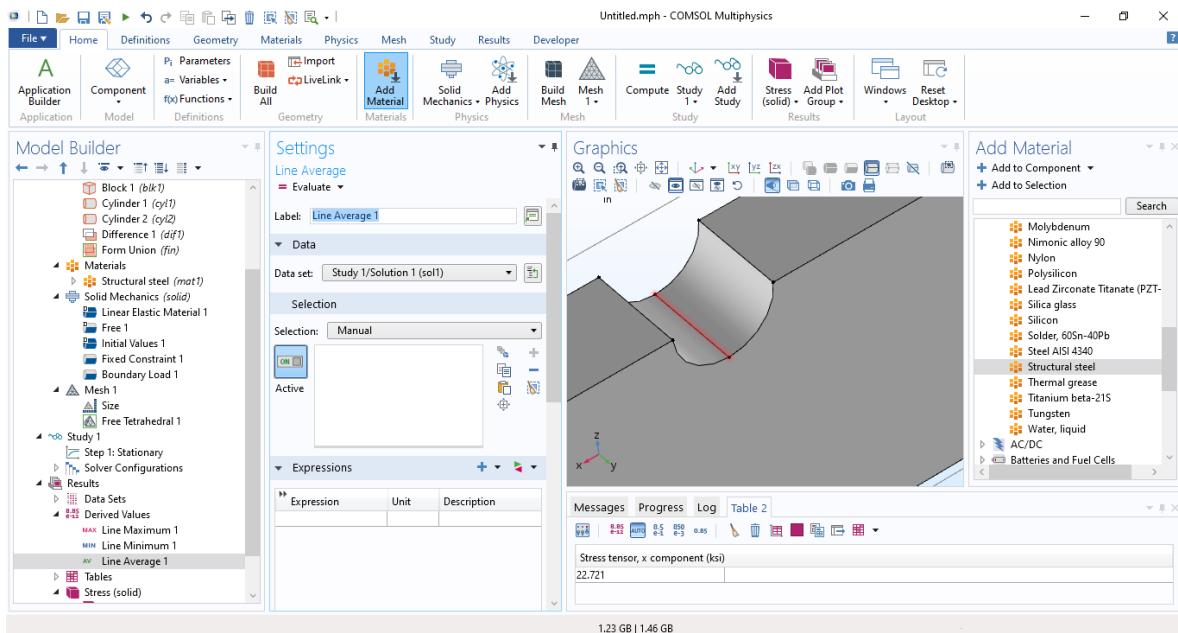
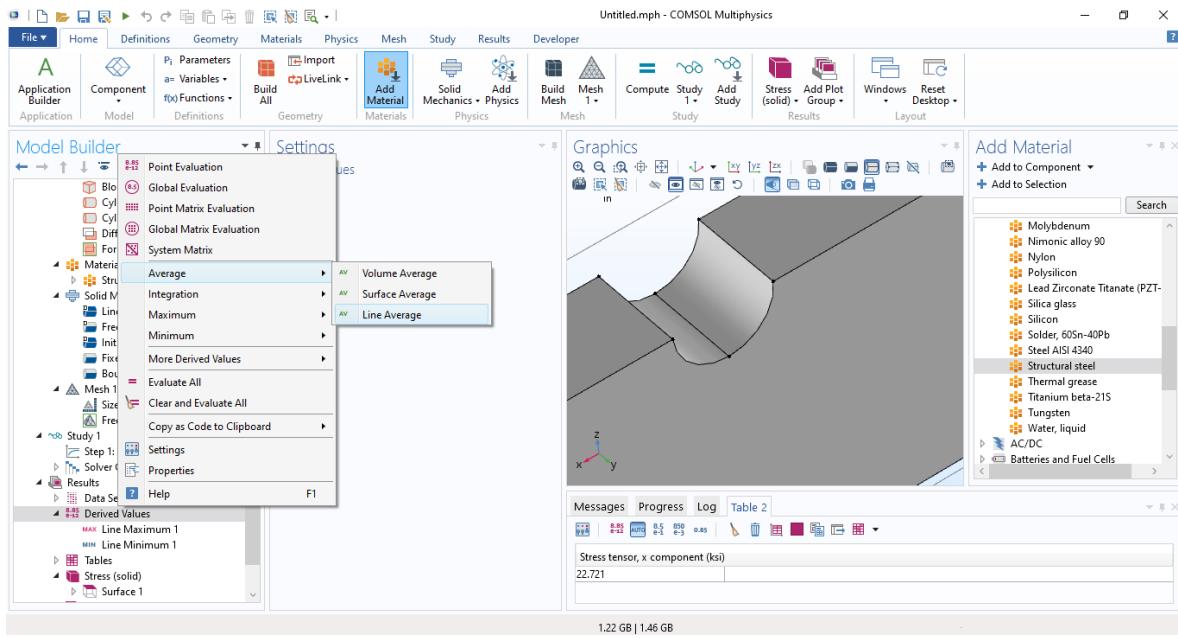


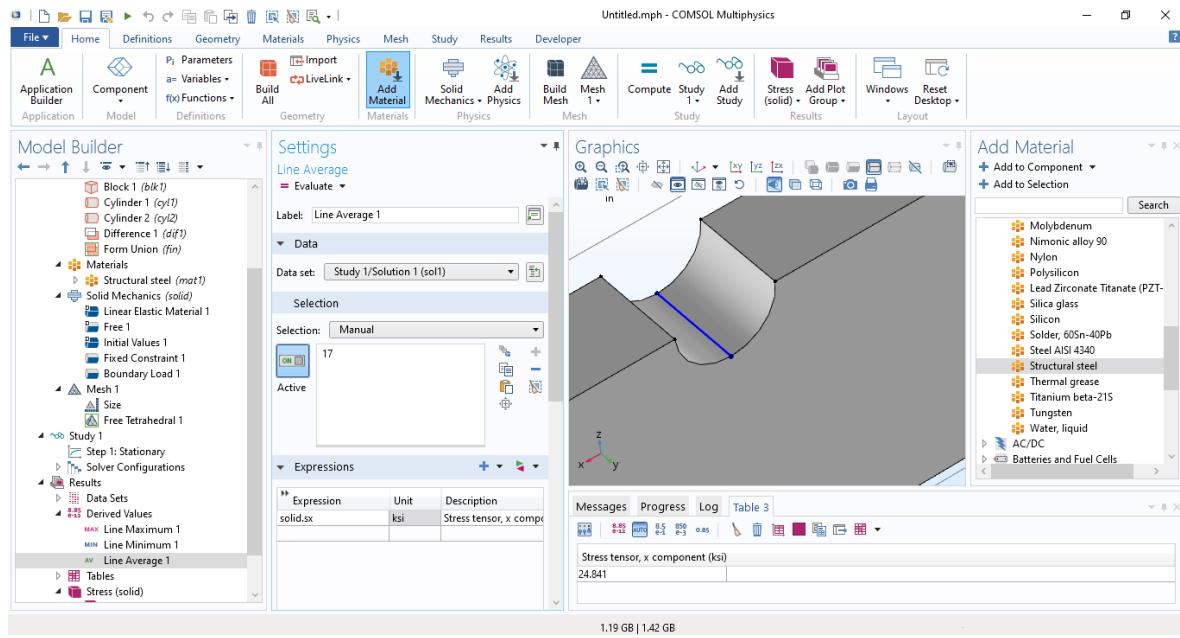








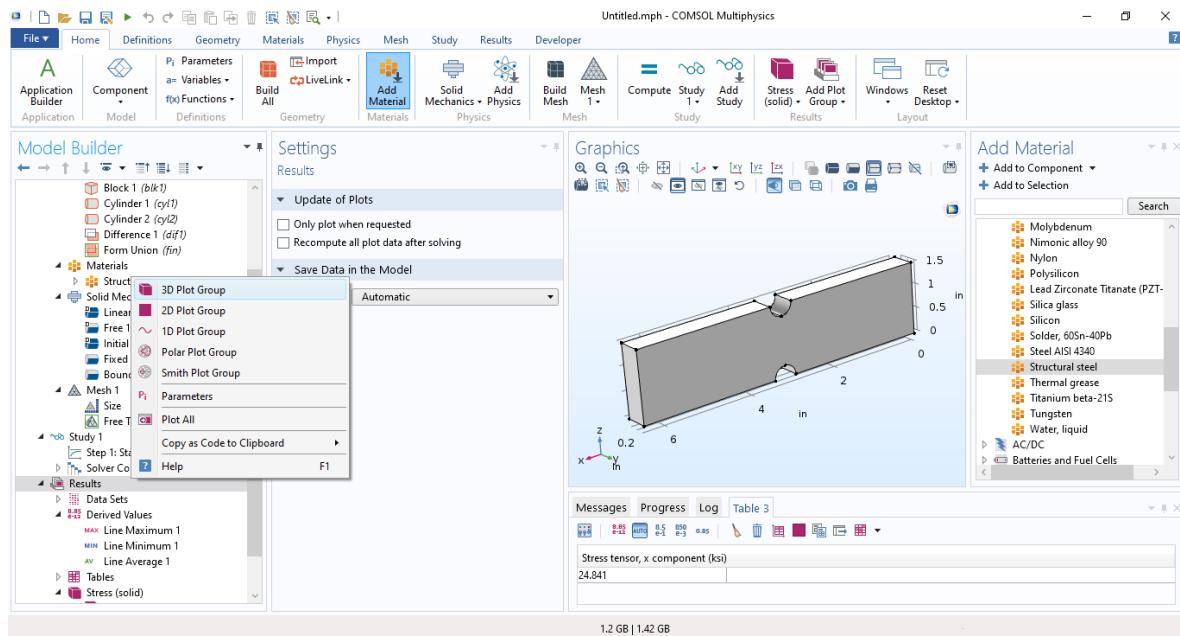


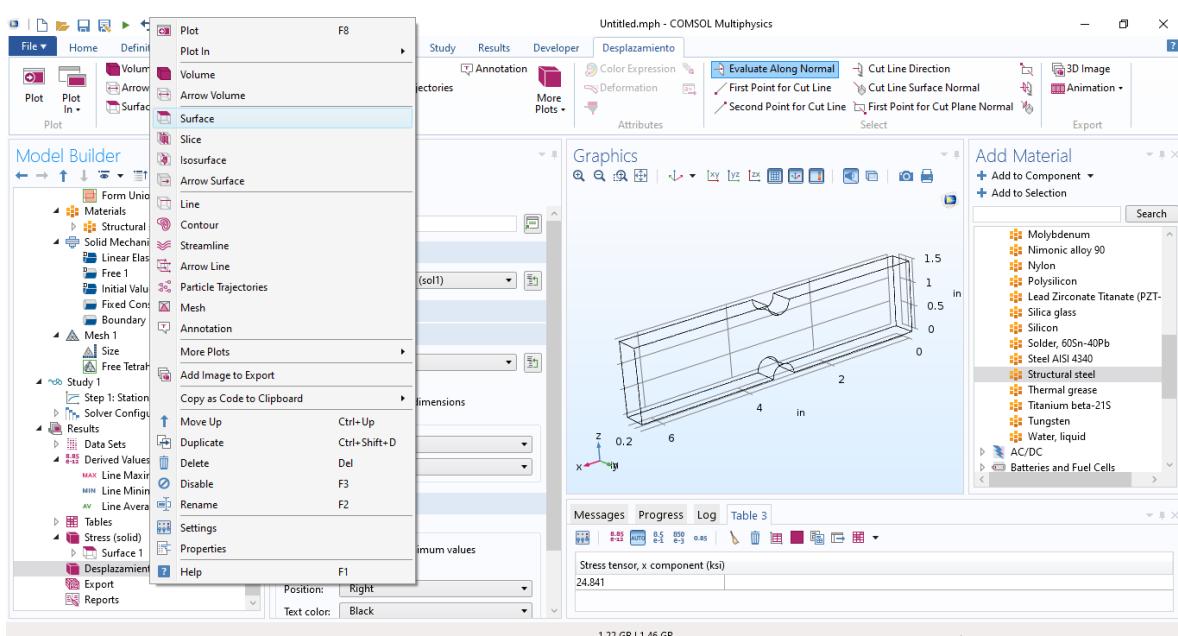
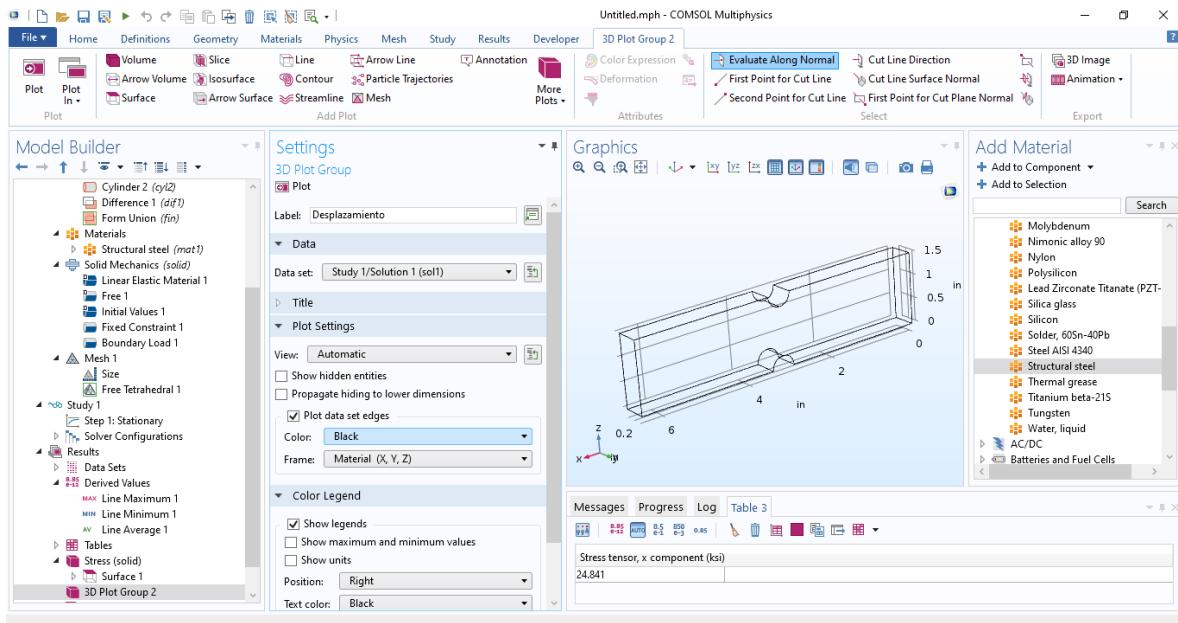


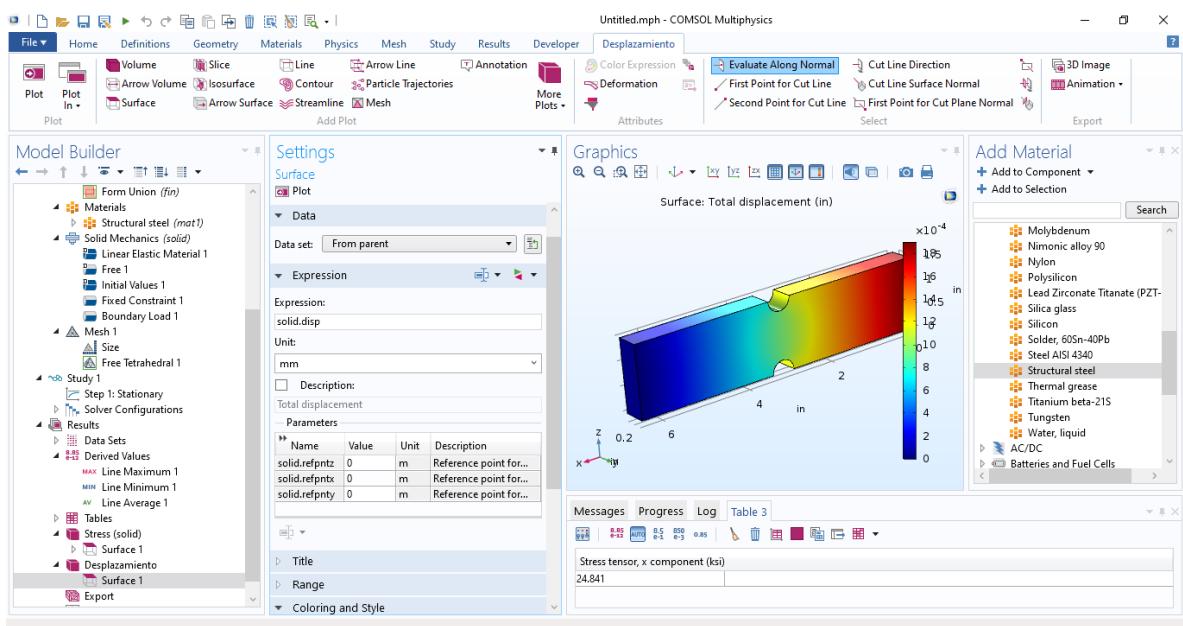
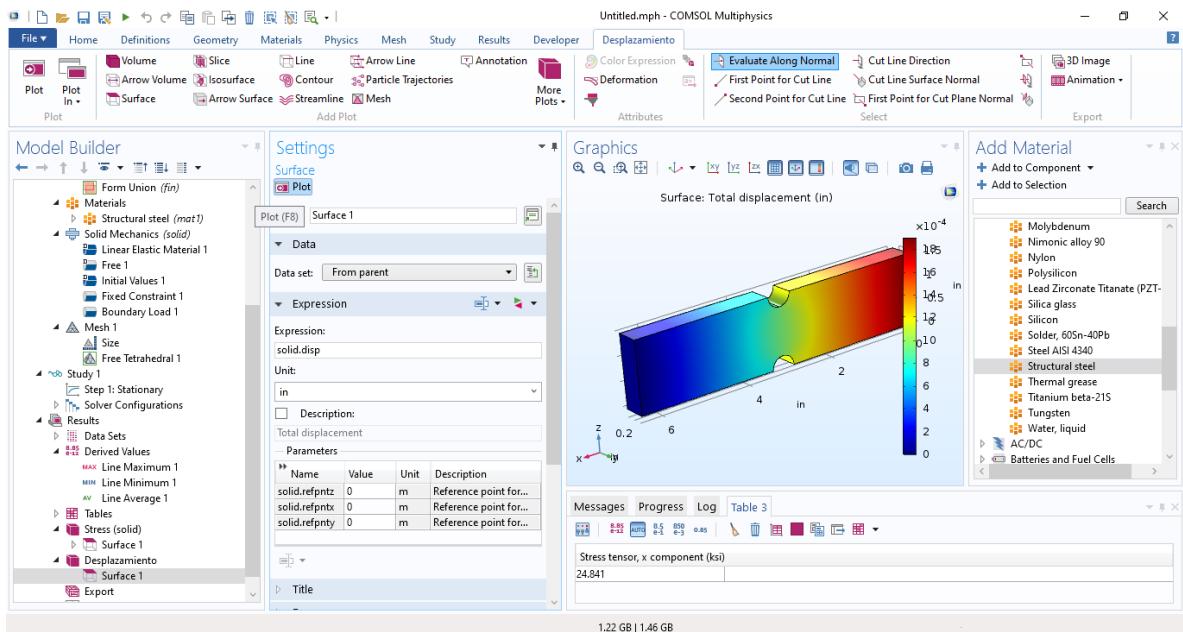
El error relativo del método analítico en comparación con el del programa se debe poner en la práctica:

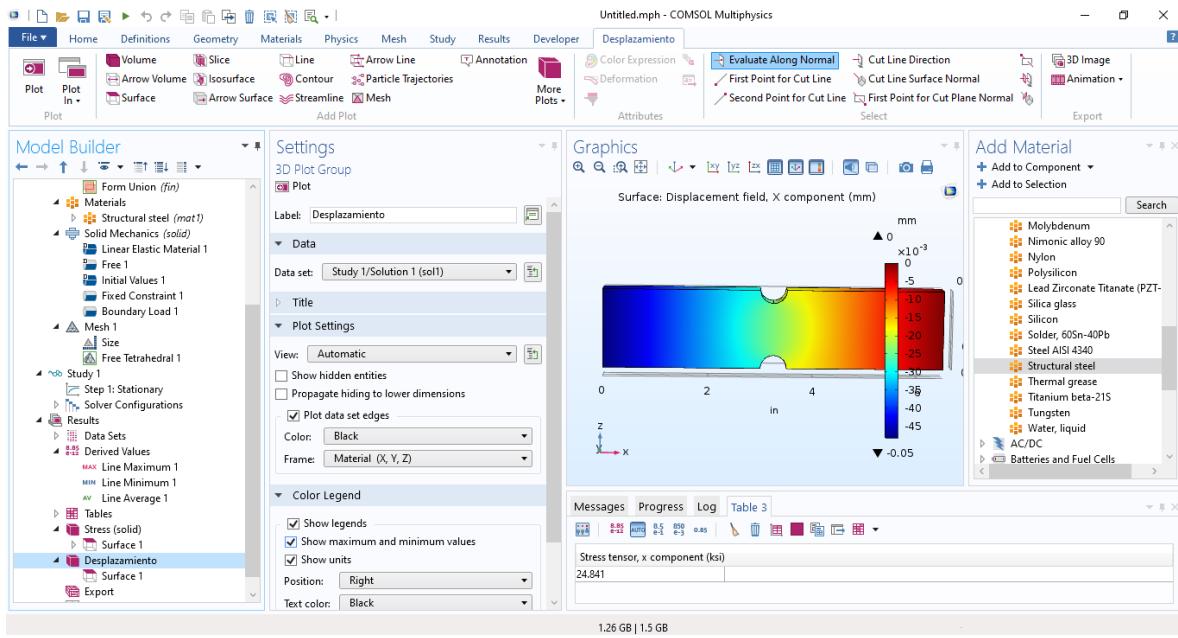
$$\text{error} = \frac{\text{valor obtenido en el programa} - \text{valor analítico}}{\text{valor obtenido en el programa}} * 100[\%]$$

El error menor al 11% es aceptable.









## BIBLIOGRAFÍA:

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