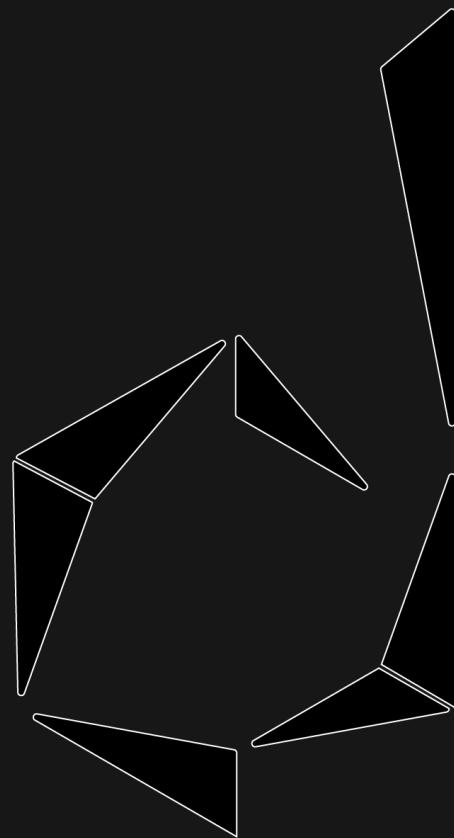


INGENIERÍA MECATRÓNICA



DI_CERO

DIEGO CERVANTES RODRÍGUEZ

INGENIERÍA ASISTIDA POR COMPUTADORA

COMSOL MULTIPHYSICS 5.6

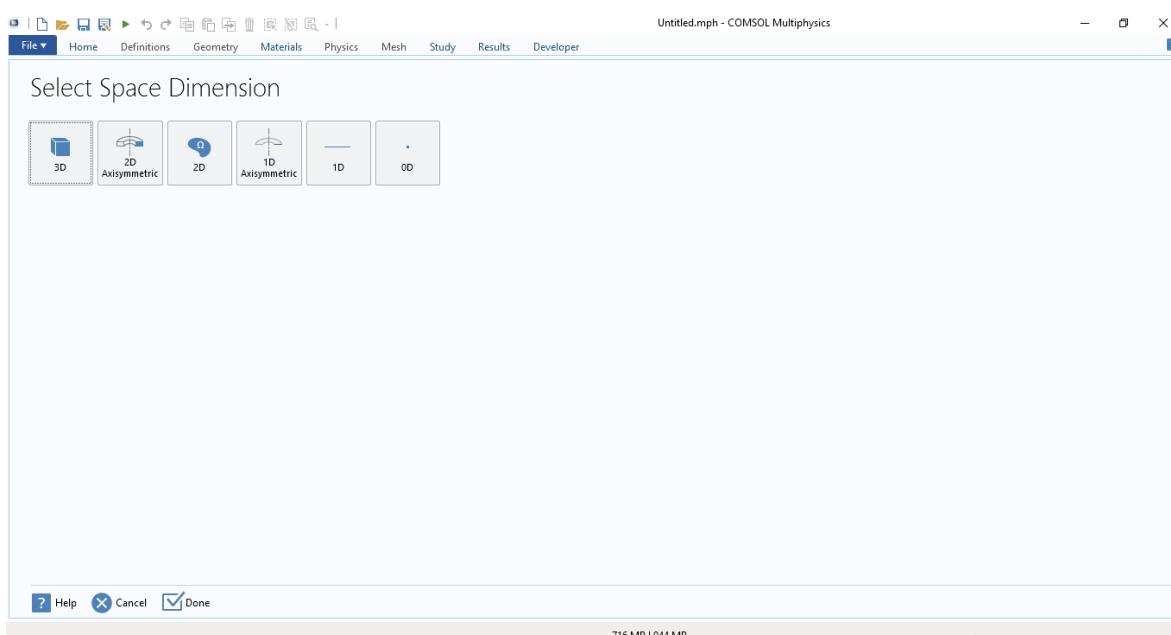
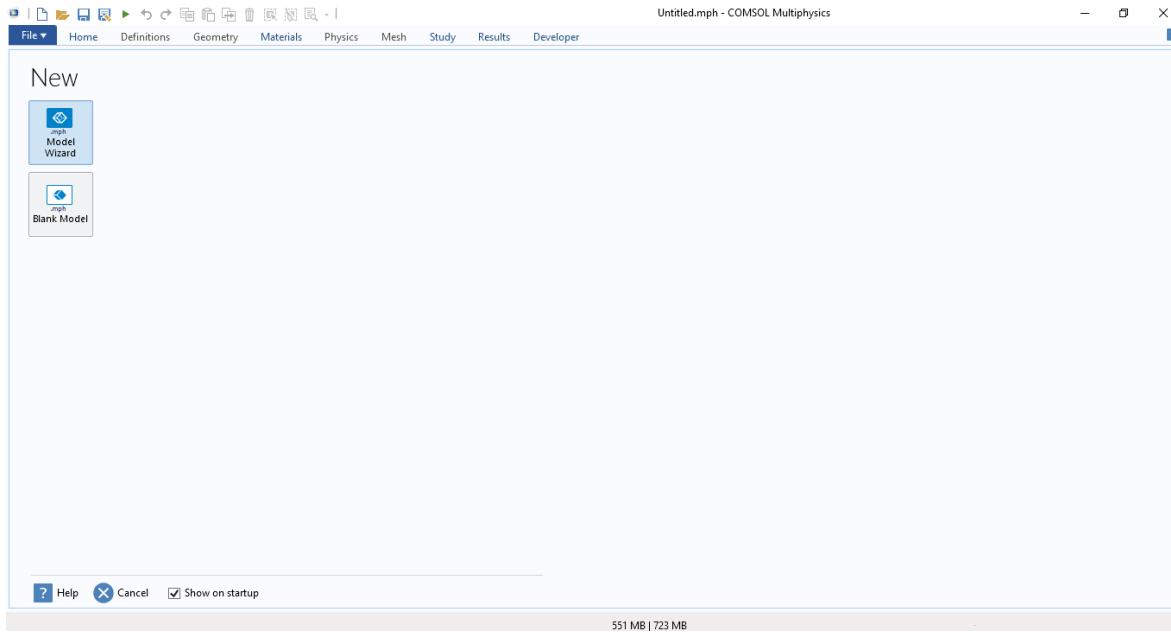
13: Torsión de una Viga con Área de Sección Transversal No Circular

Contenido

CREACIÓN DE LA PIEZA EN COMSOL:.....	2
ANÁLISIS MECÁNICO EN COMSOL:	9
RESULTADO DEL ELEMENTO FINITO EN COMSOL:	18
BIBLIOGRAFÍA:.....	26



CREACIÓN DE LA PIEZA EN COMSOL:



Select Physics

Recently Used

- Solid Mechanics (solid)
- Beam (beam)
- Truss (truss)
- AC/DC
- Acoustics
- Chemical Species Transport
- Electrochemistry
- Fluid Flow
- Heat Transfer
- Optics
- Plasma
- Radio Frequency
- Semiconductor
- Structural Mechanics

Add

Added physics interfaces:

Space Dimension Study

? Help X Cancel Done

736 MB | 970 MB

Select Study

Preset Studies

- Bolt Pre-Tension
- Eigenfrequency
- Frequency Domain
- Frequency Domain Modal
- Linear Buckling
- Modal Reduced-Order Model
- Prestressed Analysis, Eigenfrequency
- Prestressed Analysis, Frequency Domain
- Stationary
- Time Dependent

Added study:

- Stationary

Added physics interfaces:

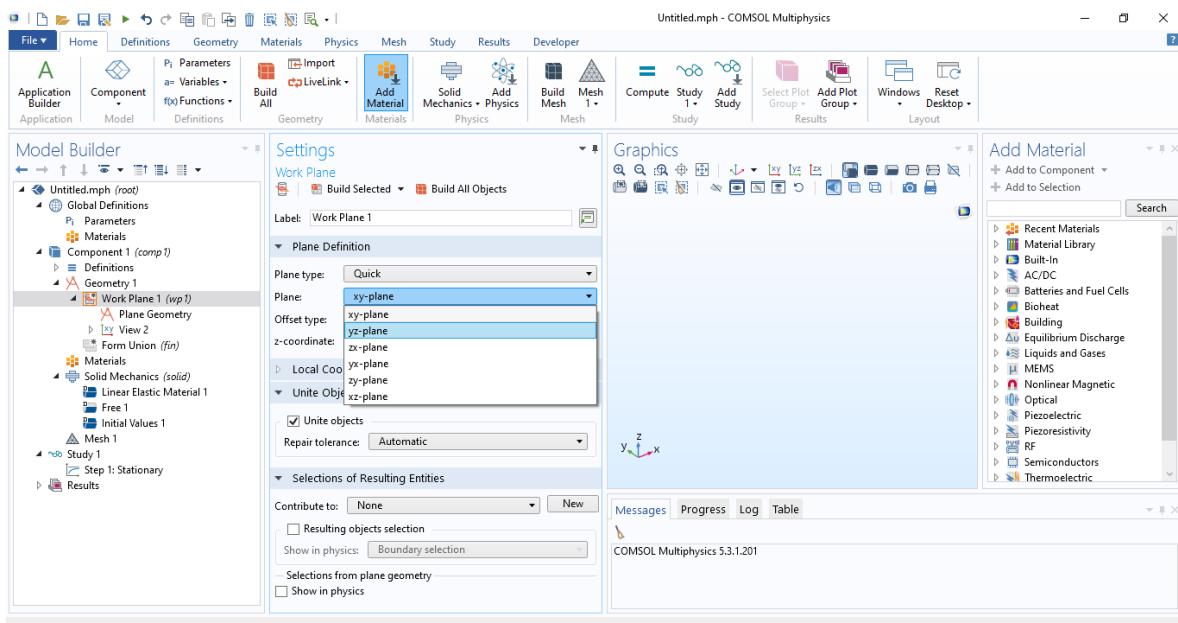
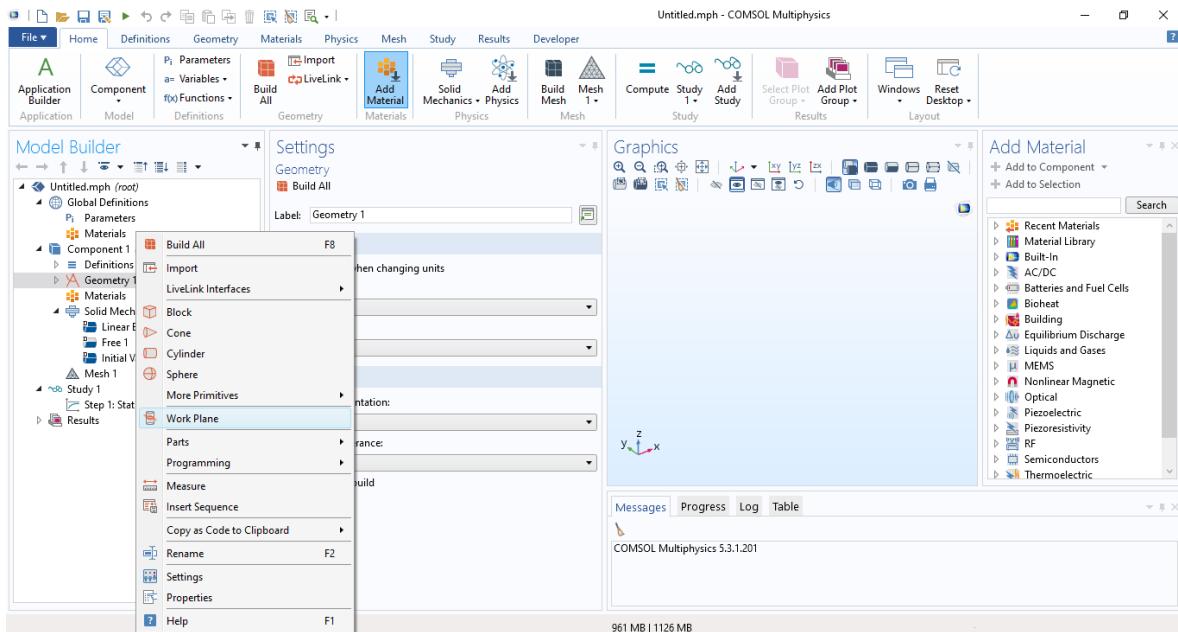
- Solid Mechanics (solid)

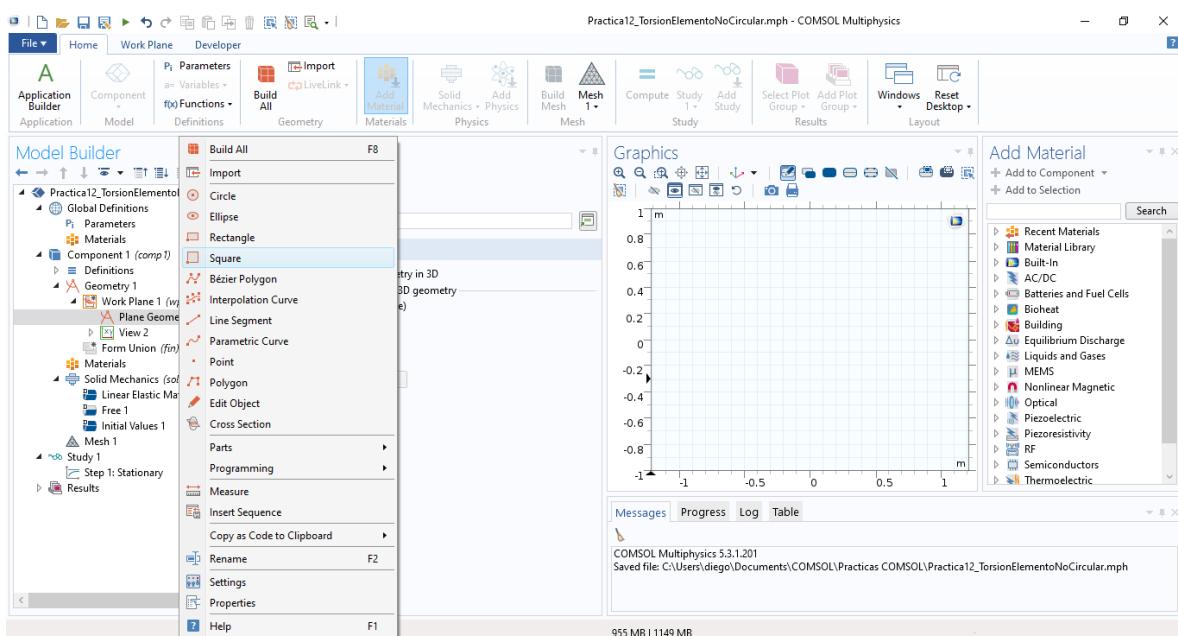
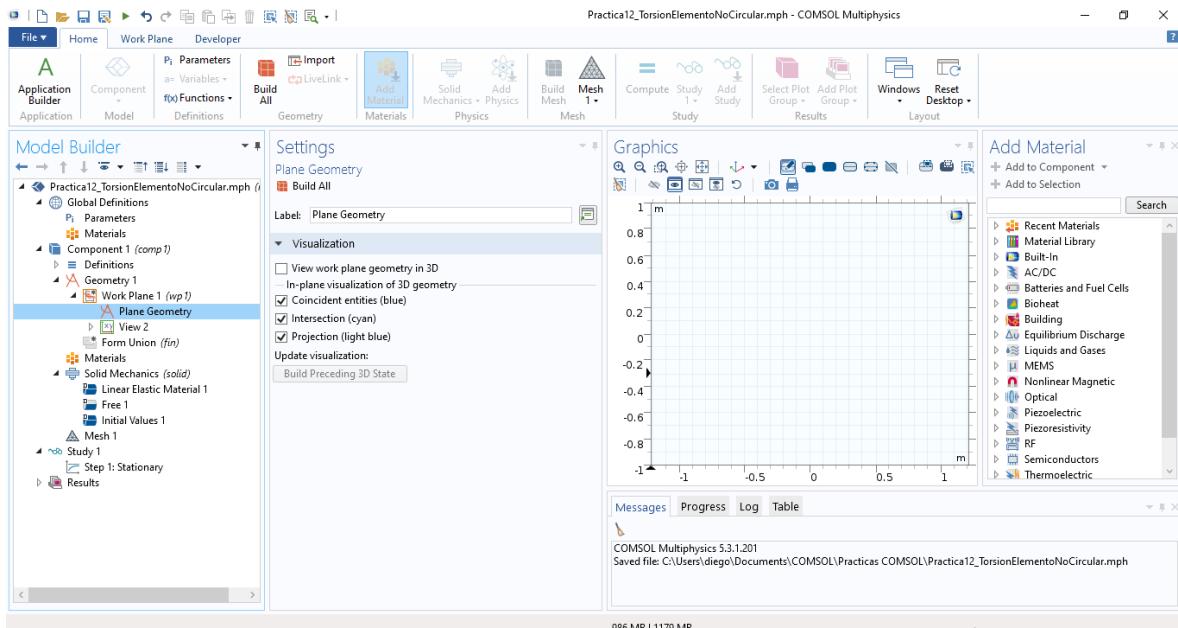
Physics

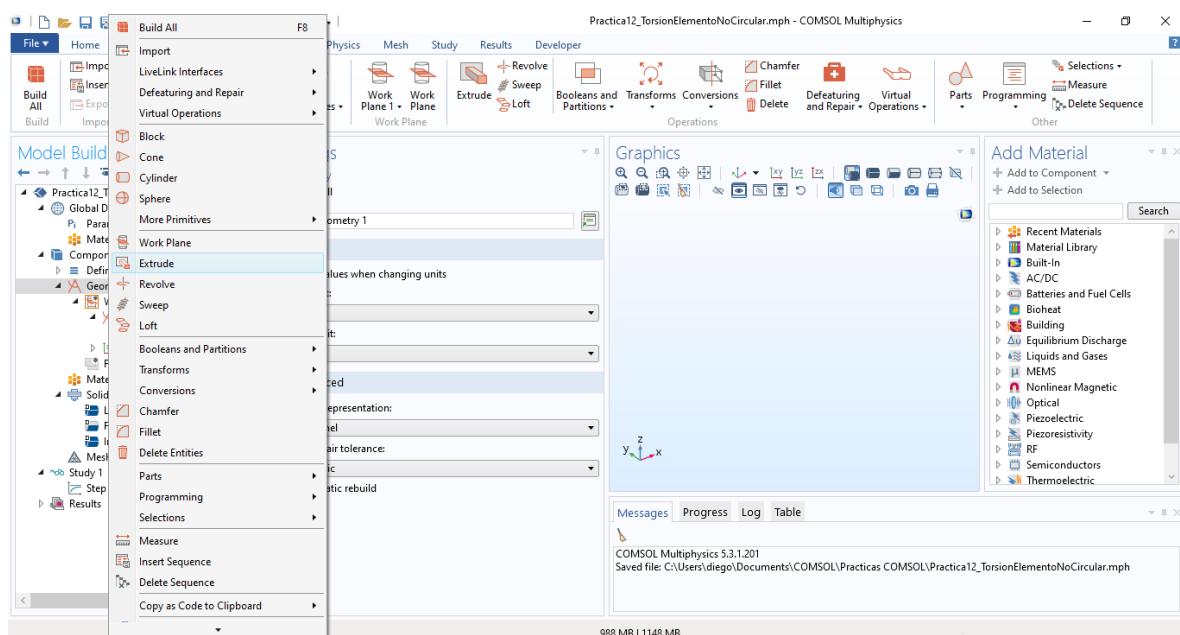
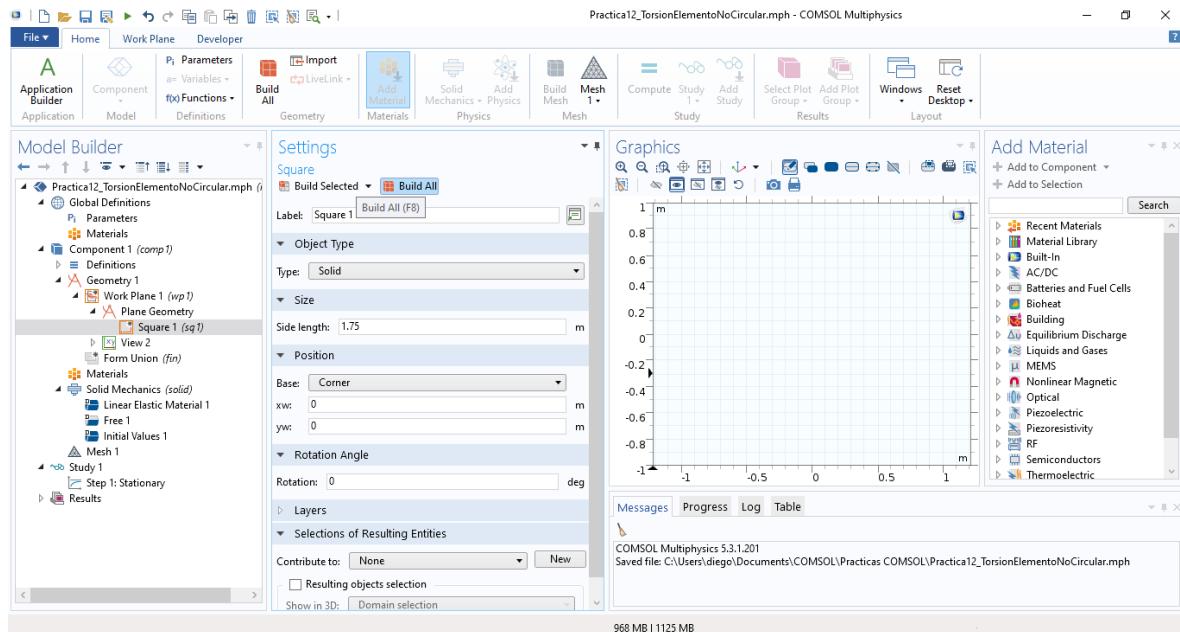
? Help X Cancel Done

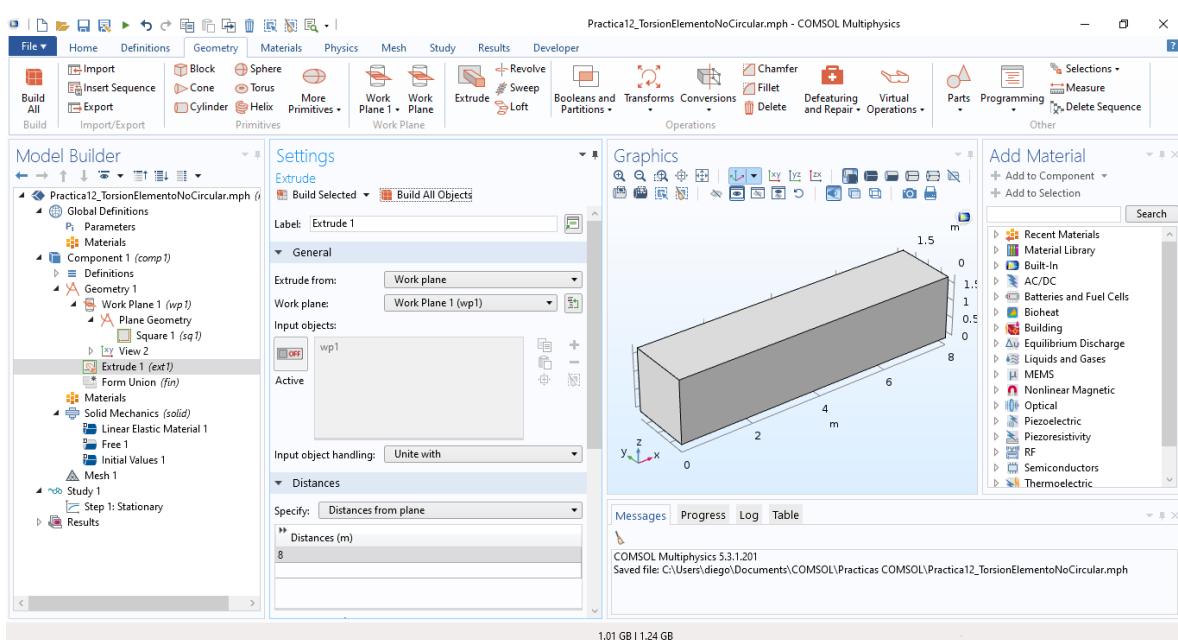
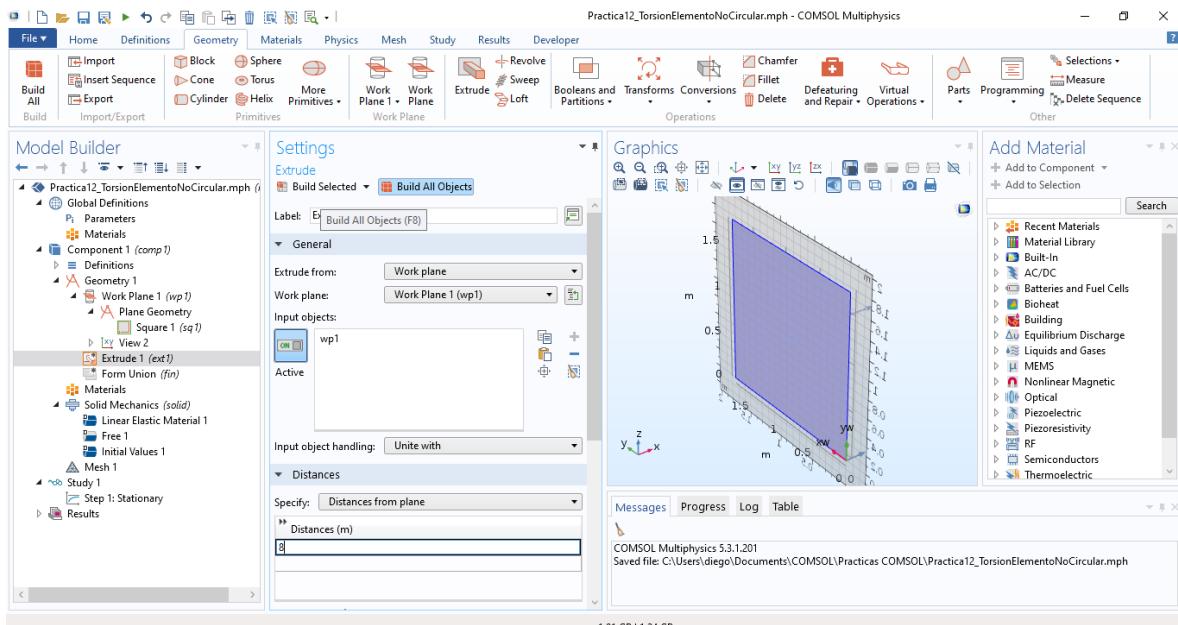
Add Selected Study and Physics Interfaces and Finish

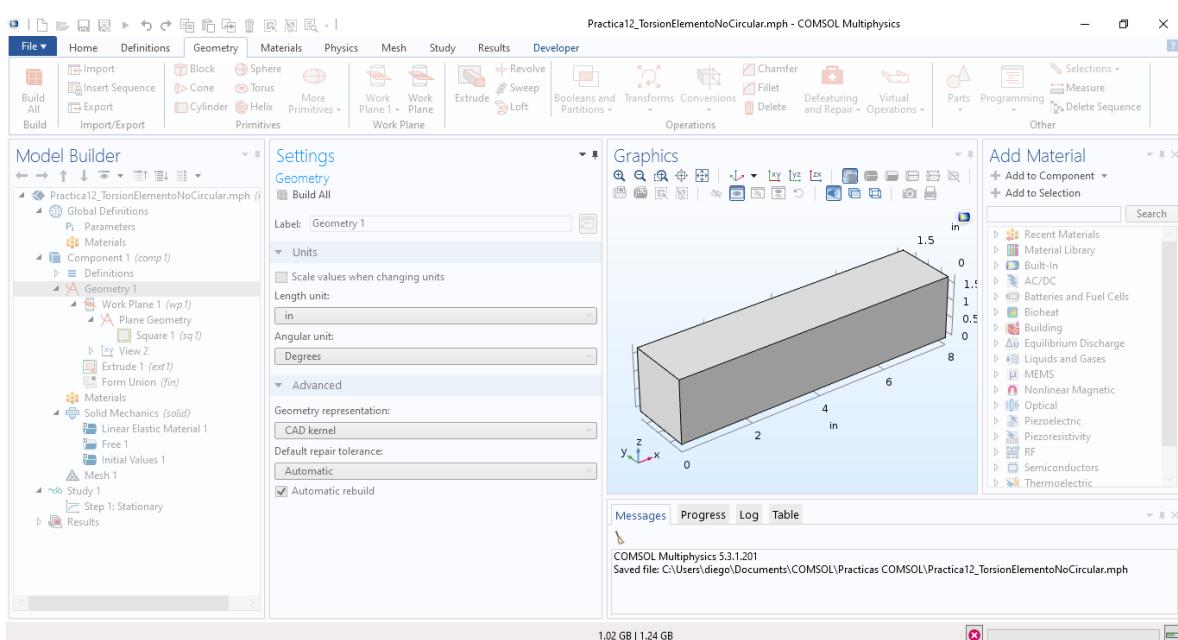
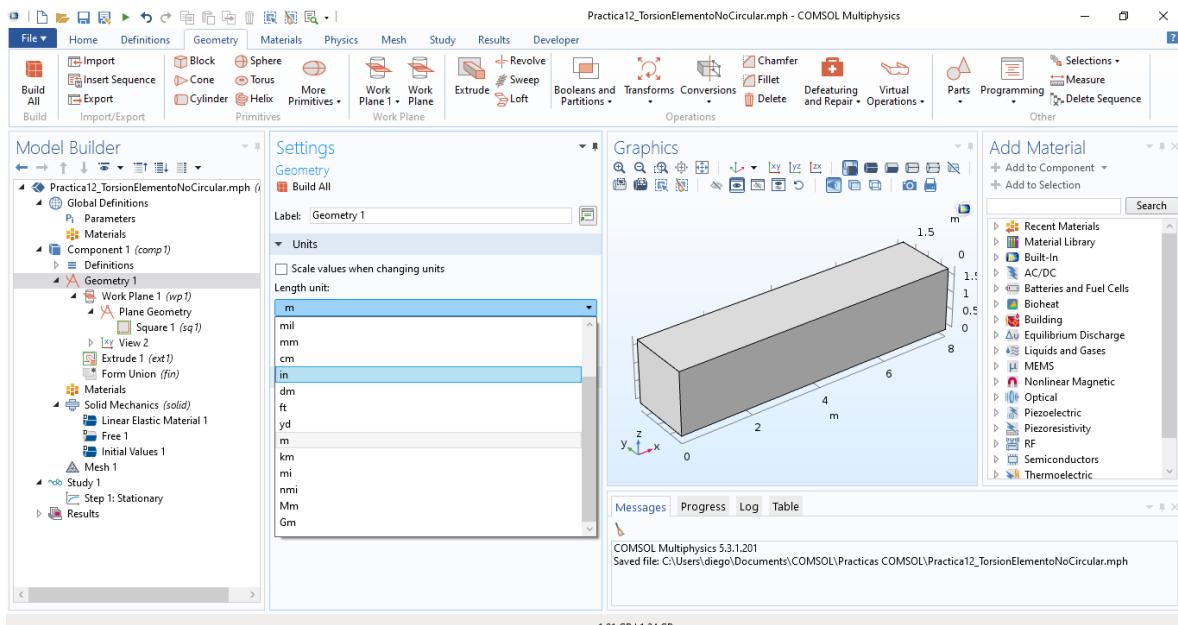
838 MB | 1060 MB



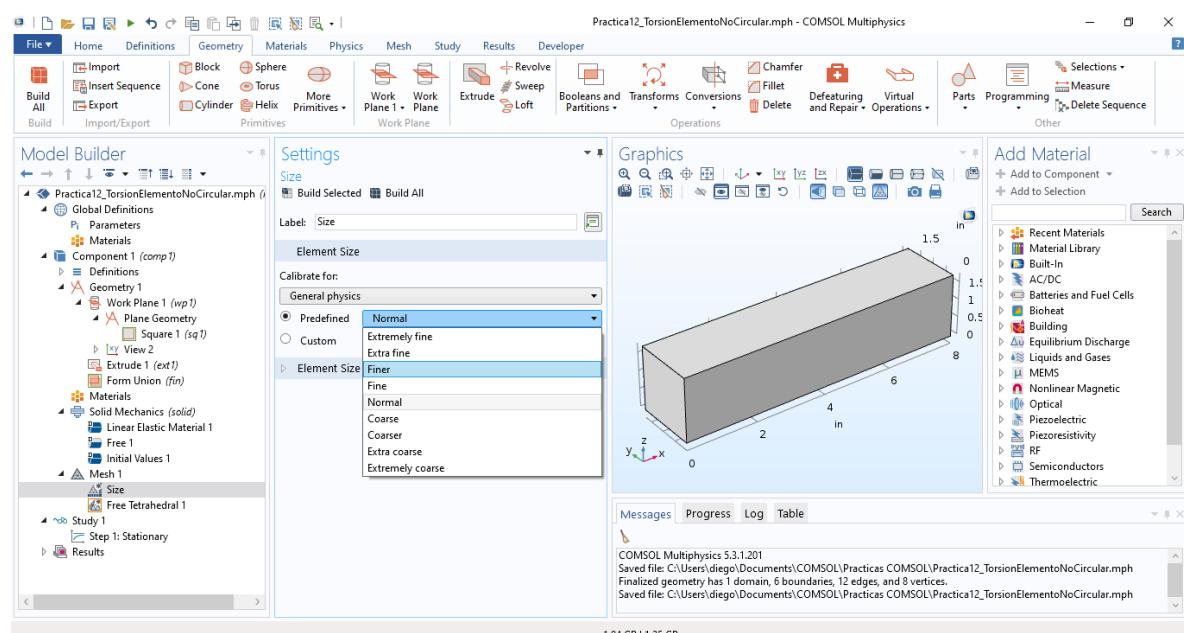
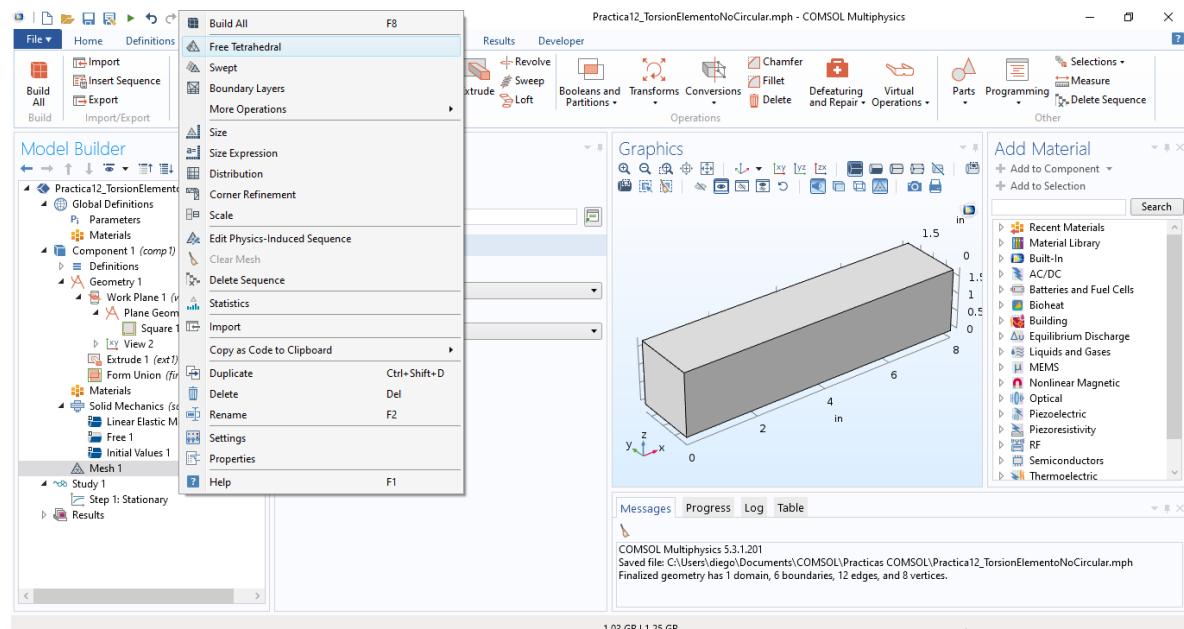


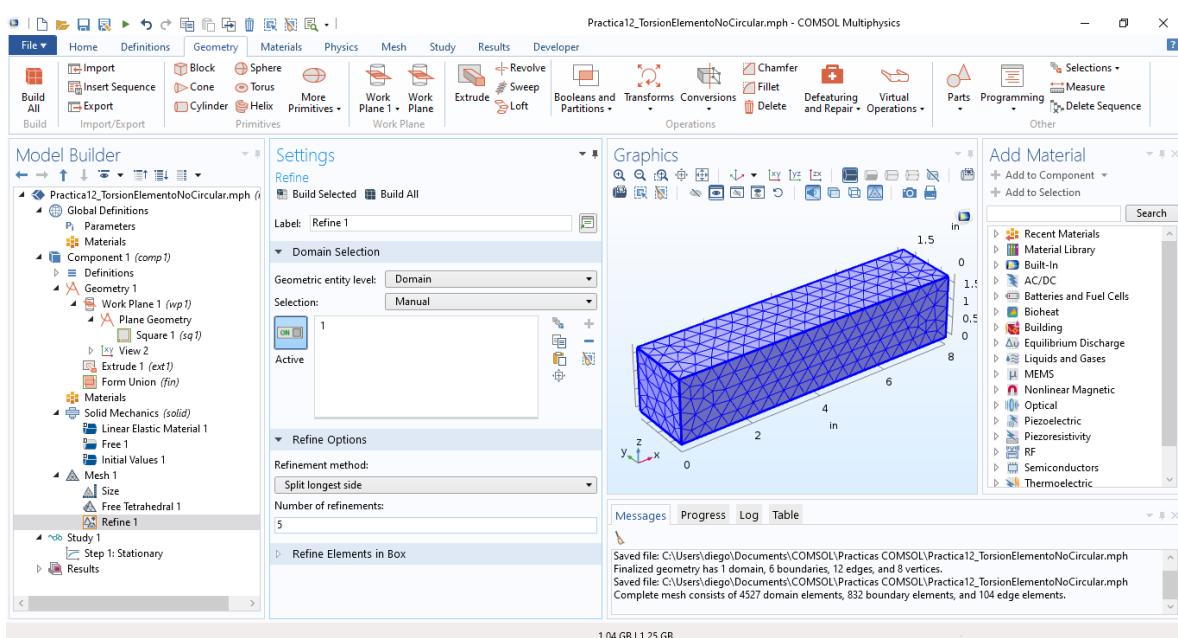
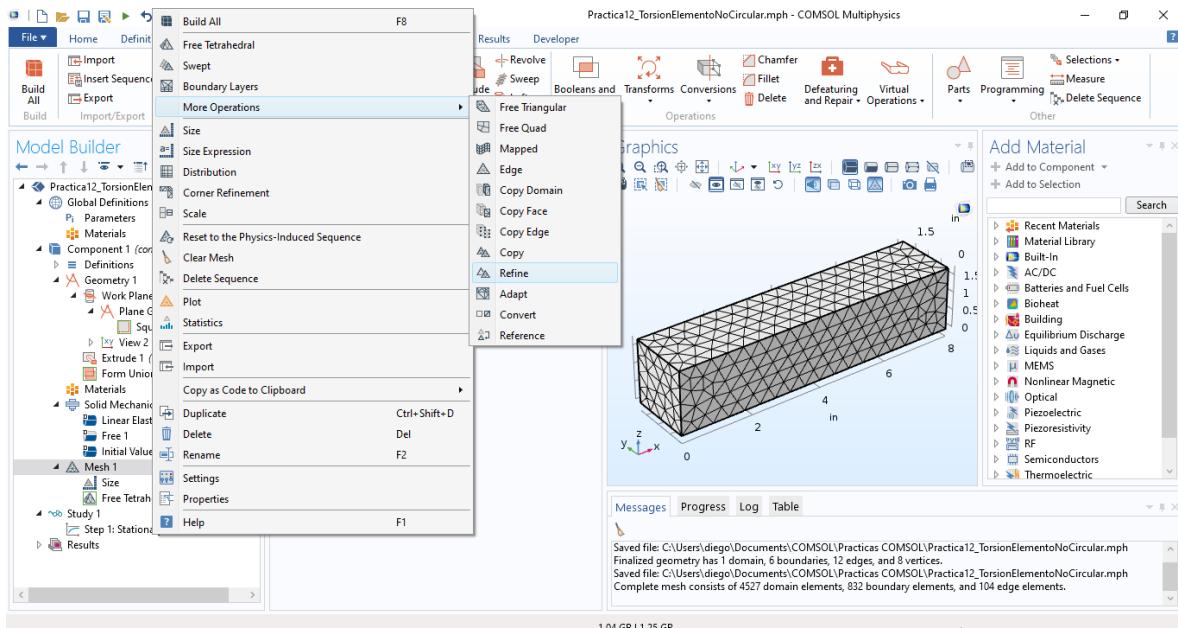


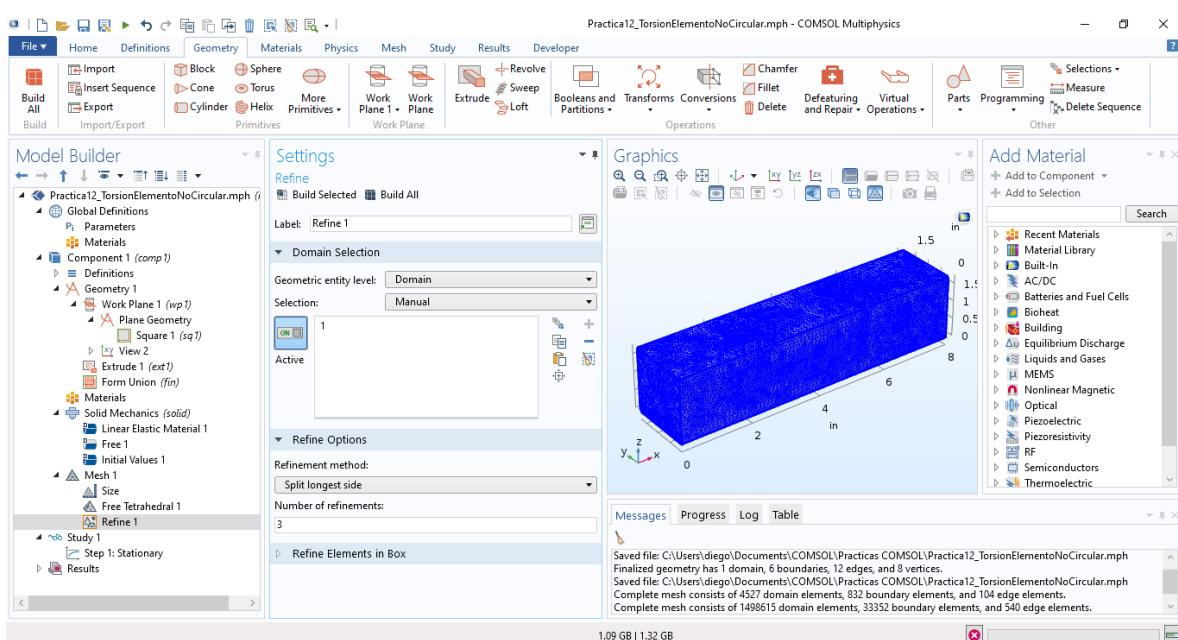
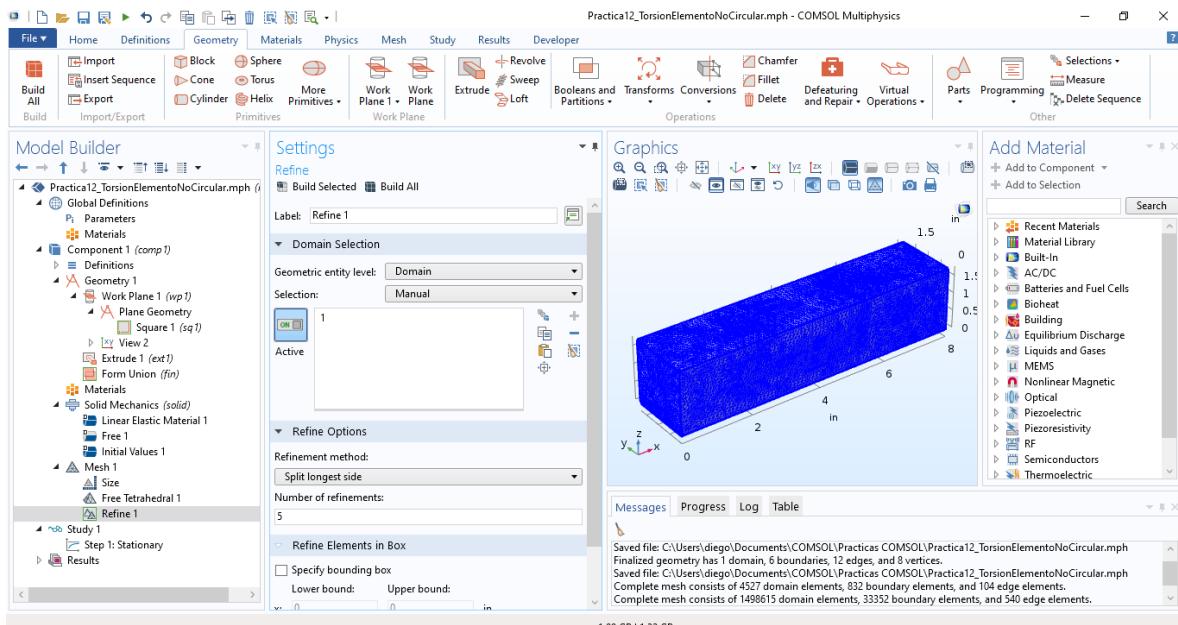


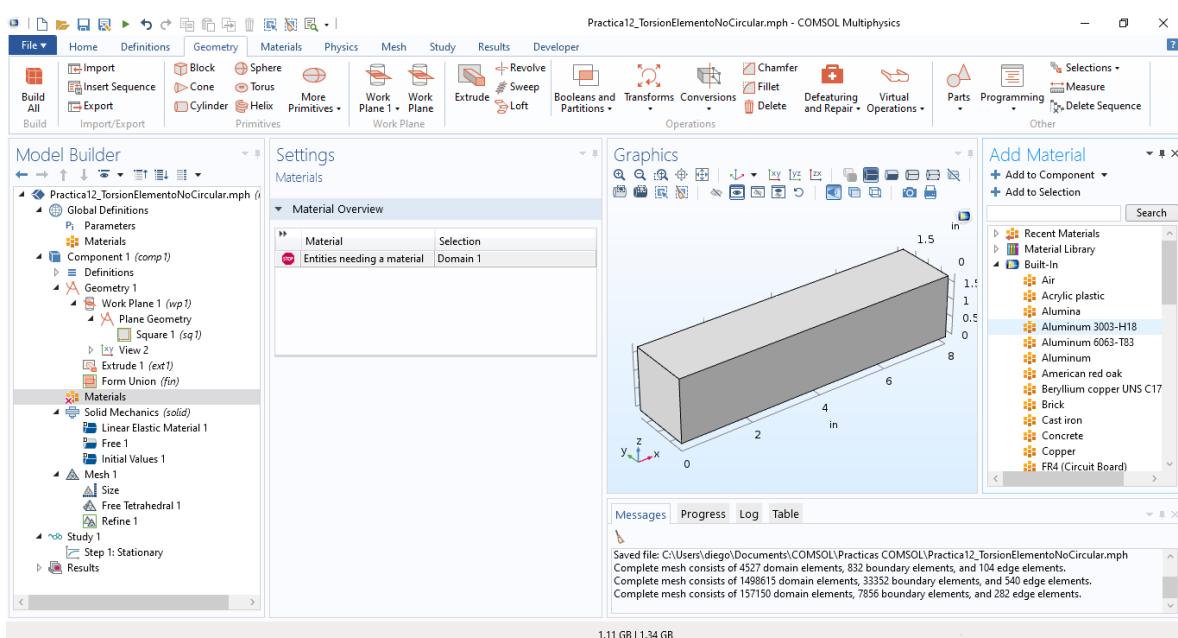
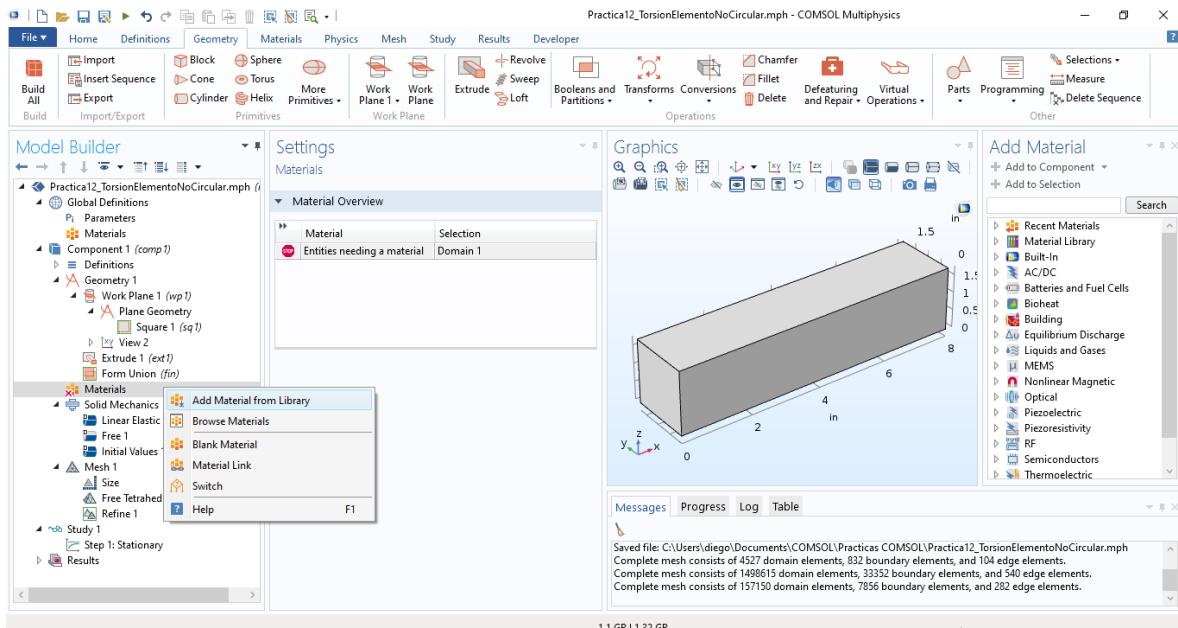


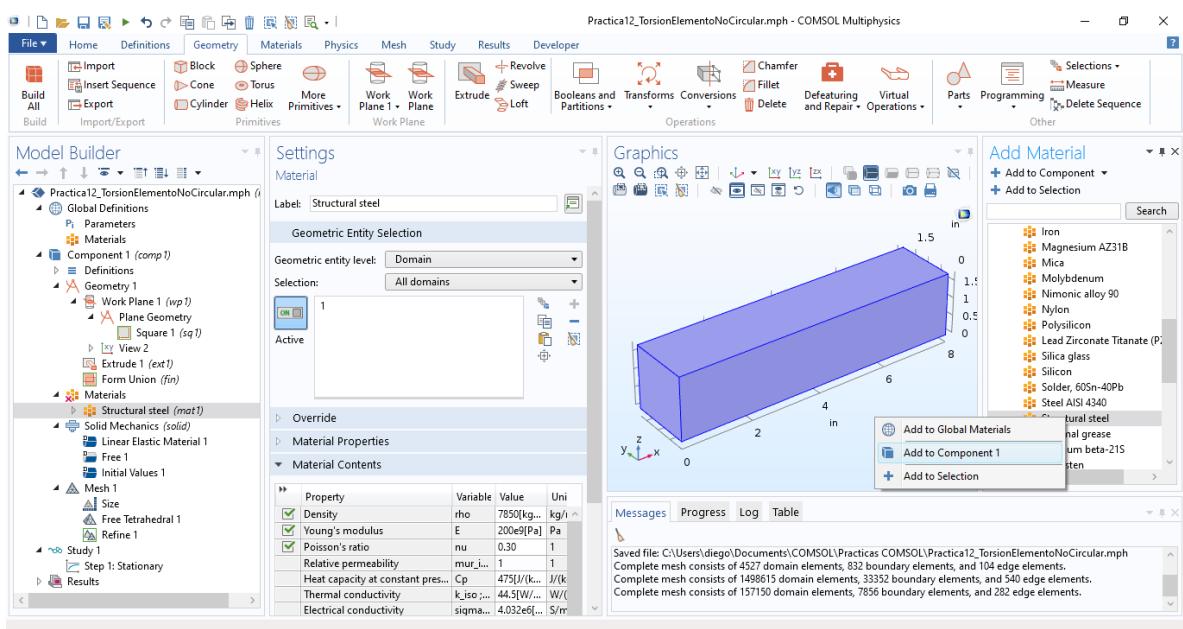
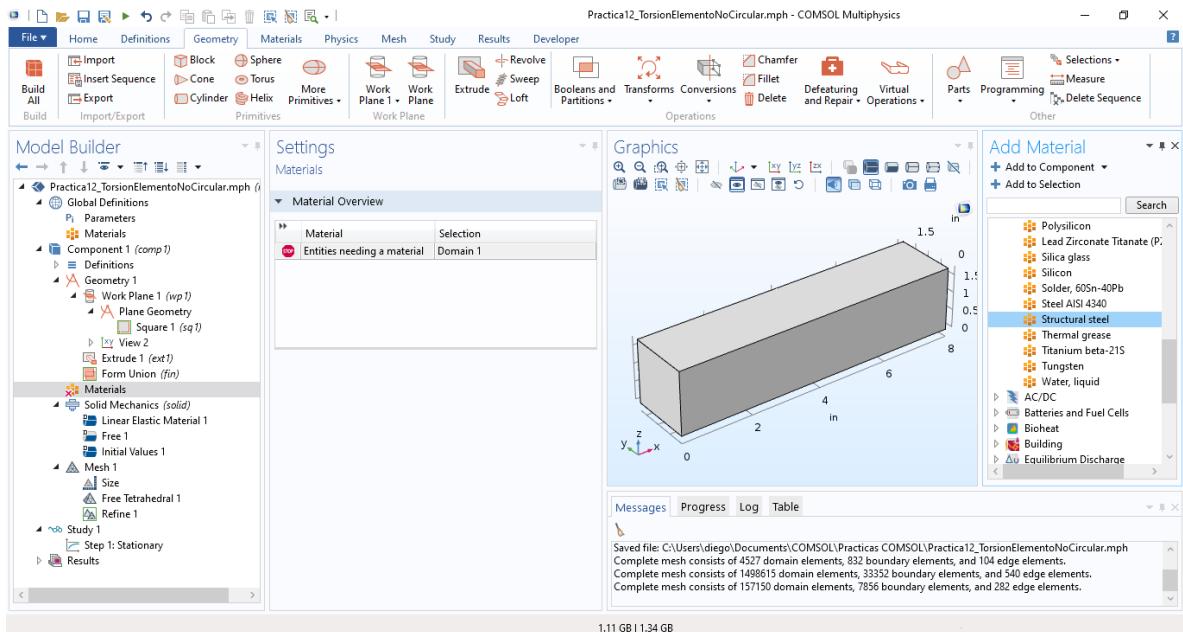
ANÁLISIS MECÁNICO EN COMSOL:

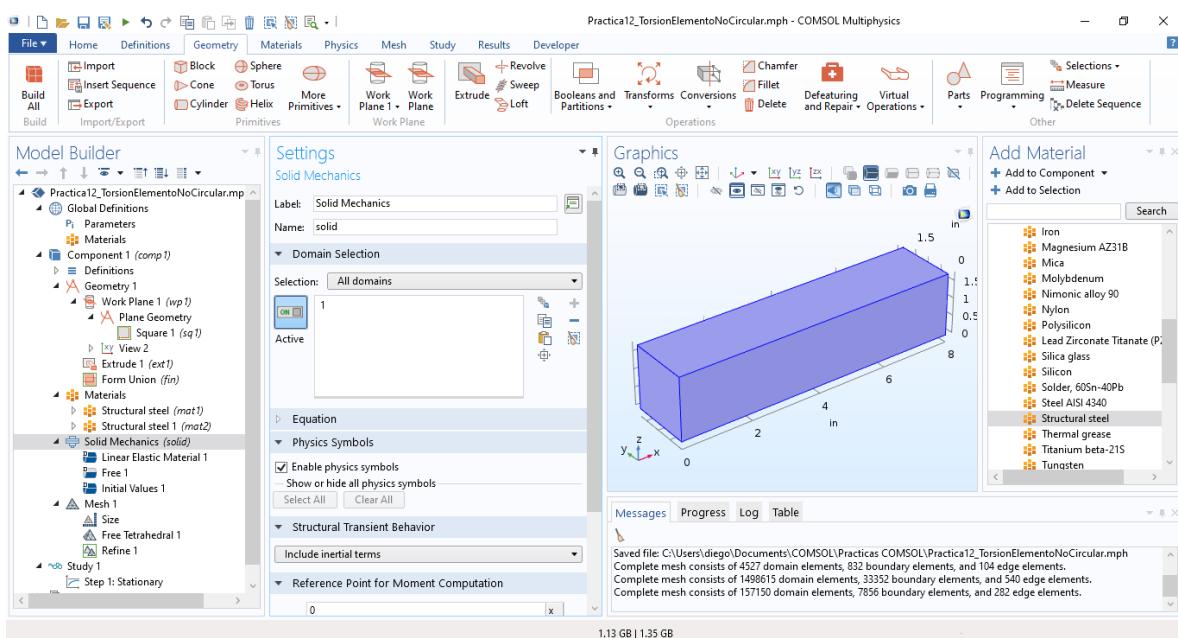
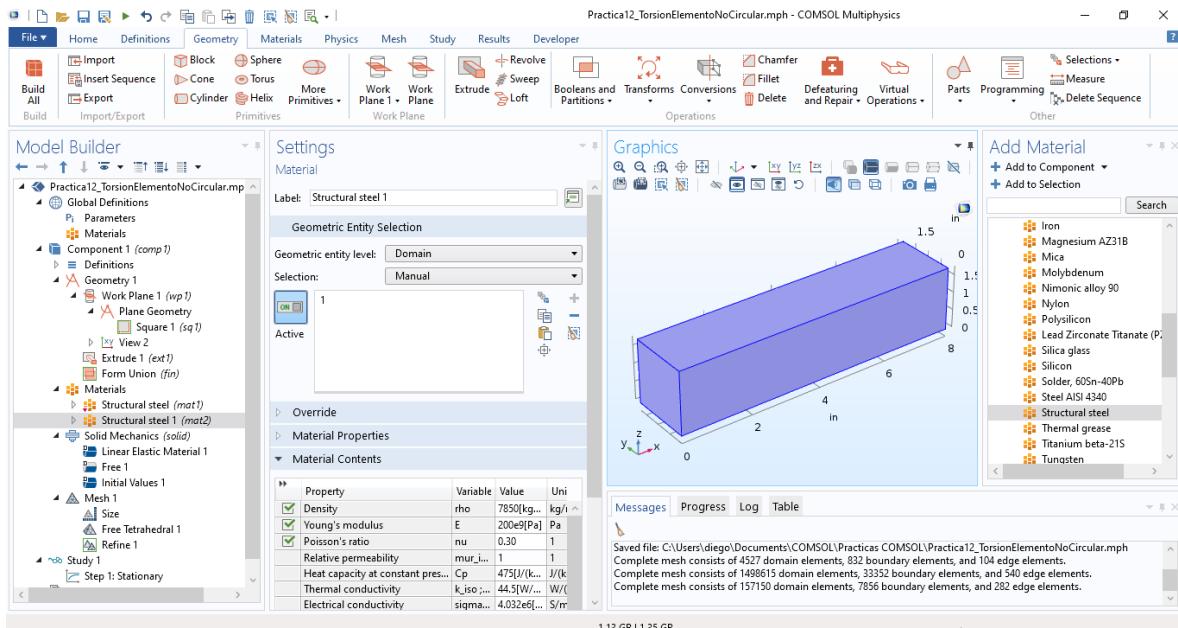


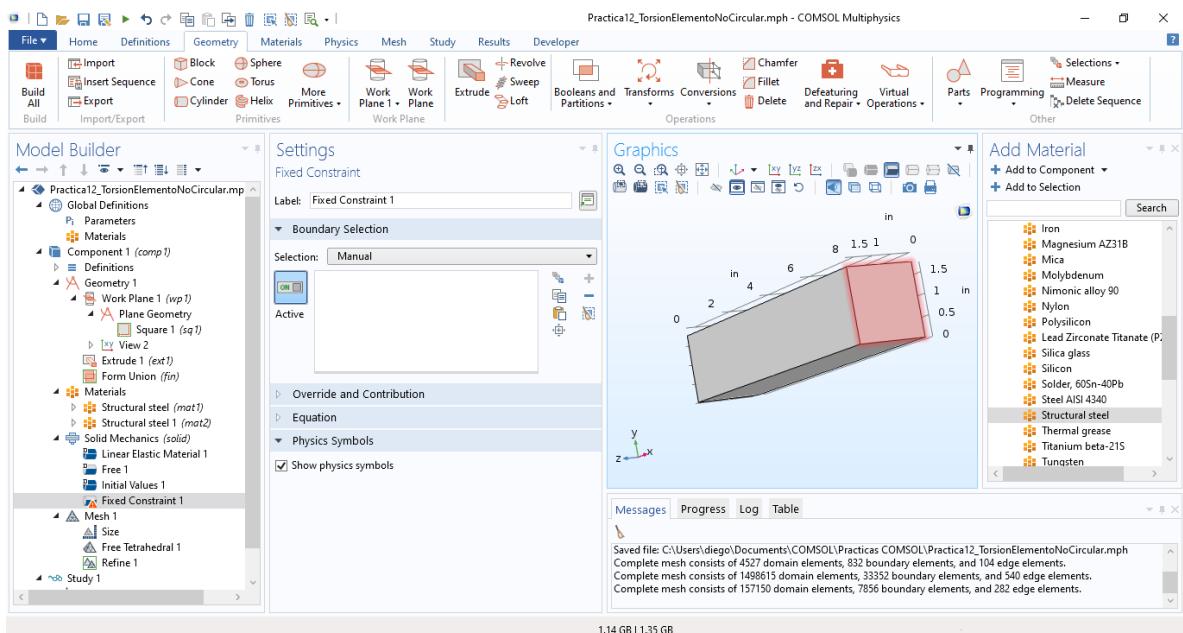
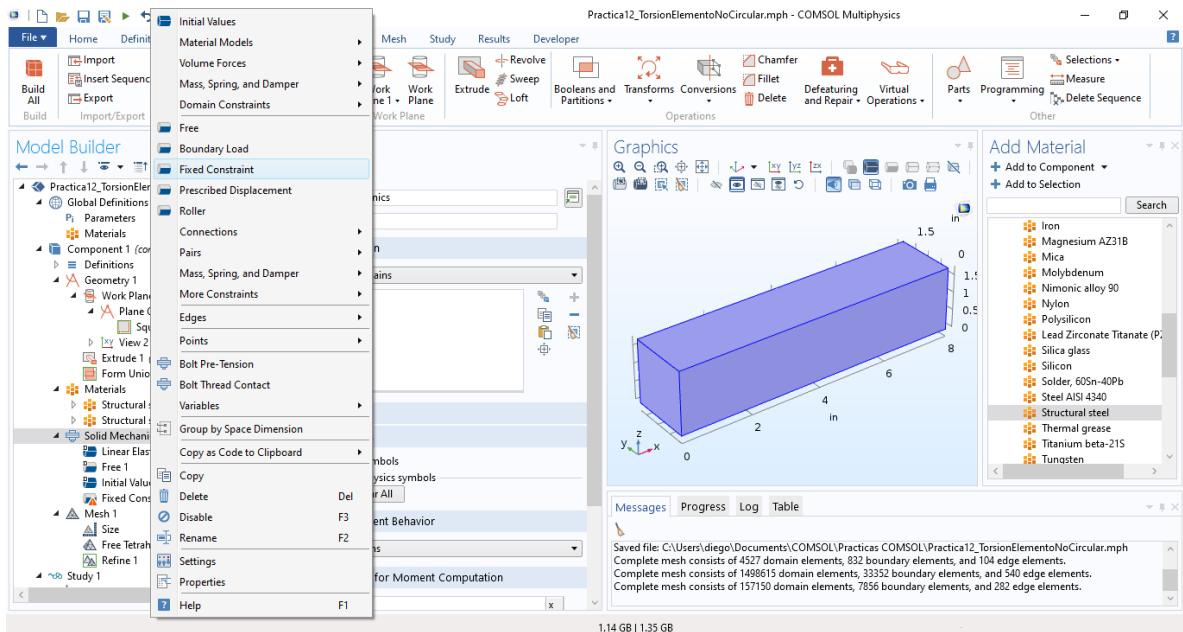


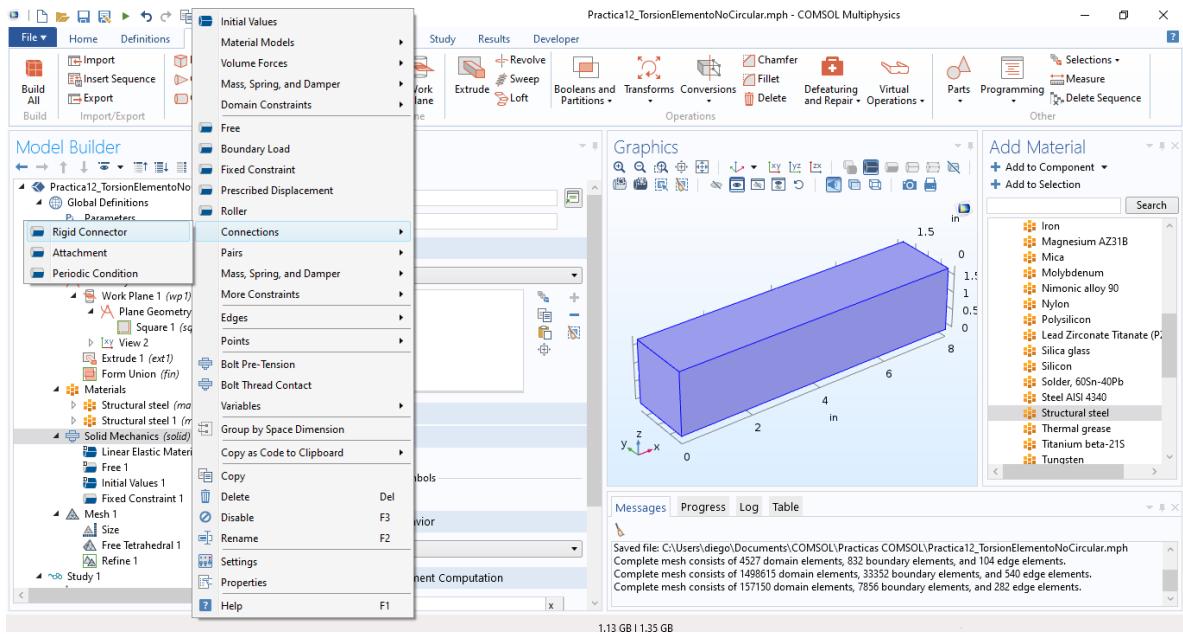




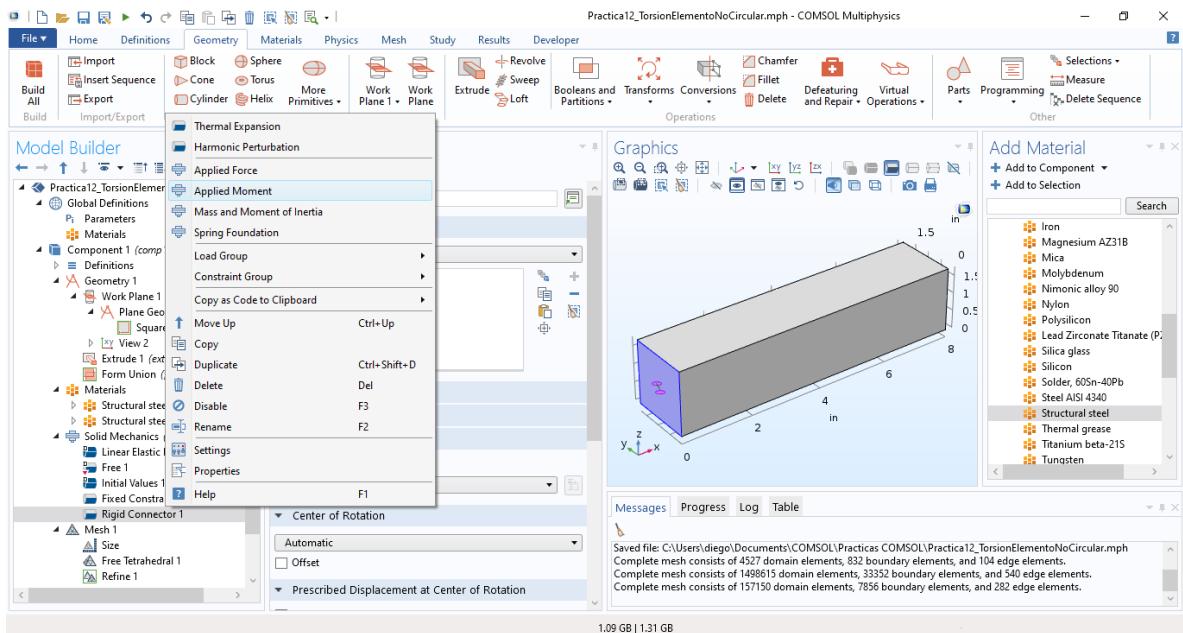


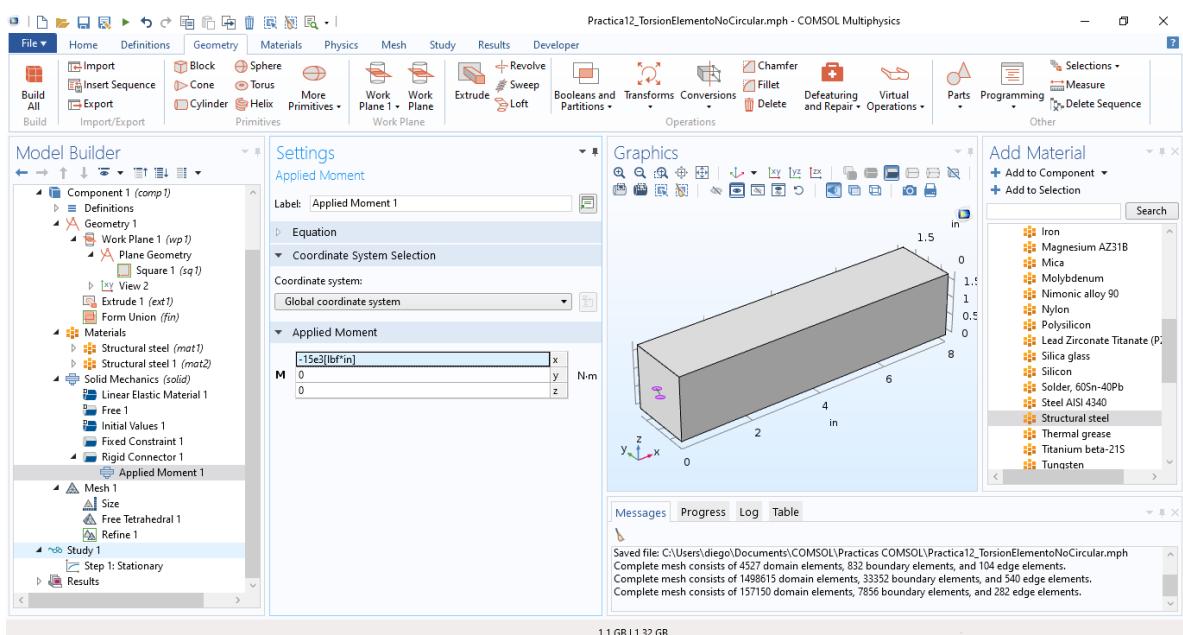
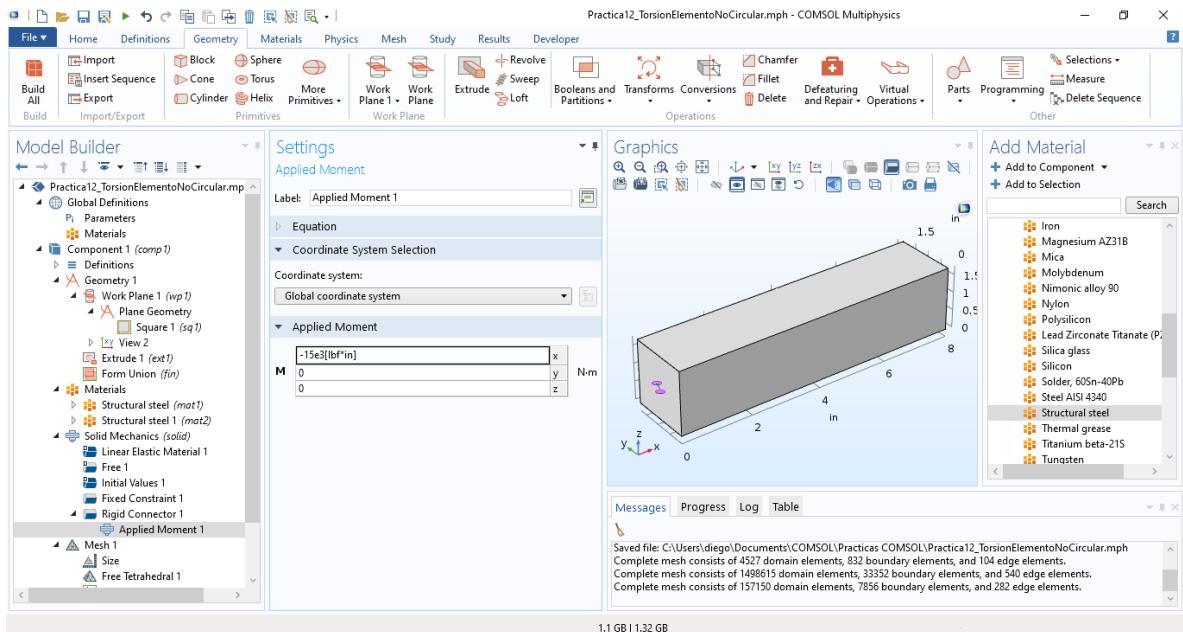




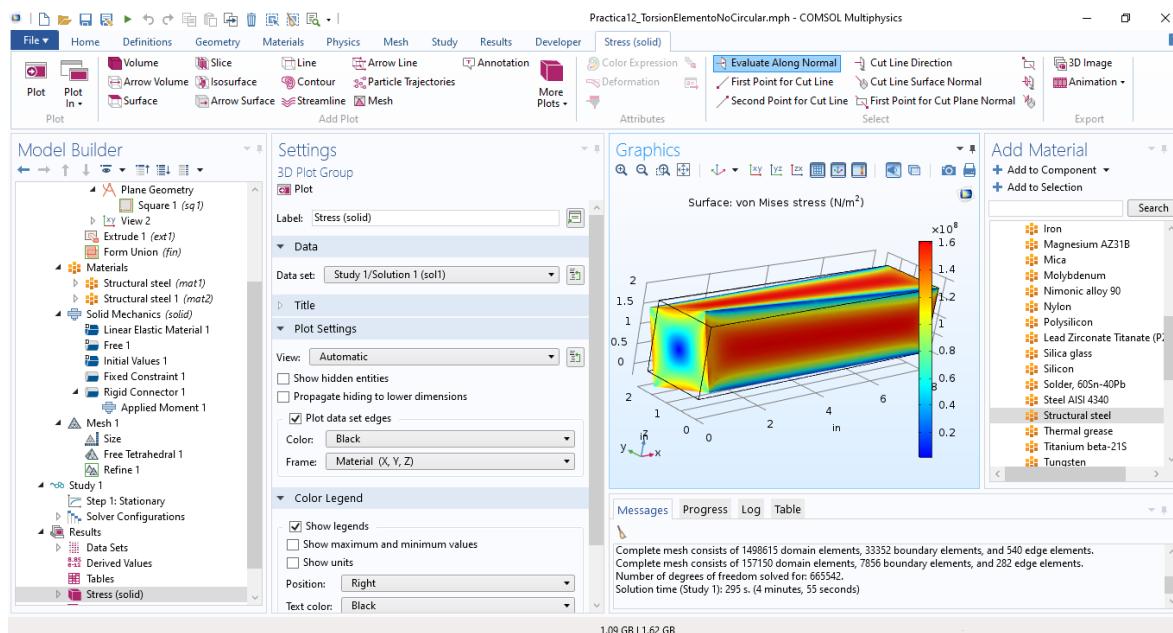
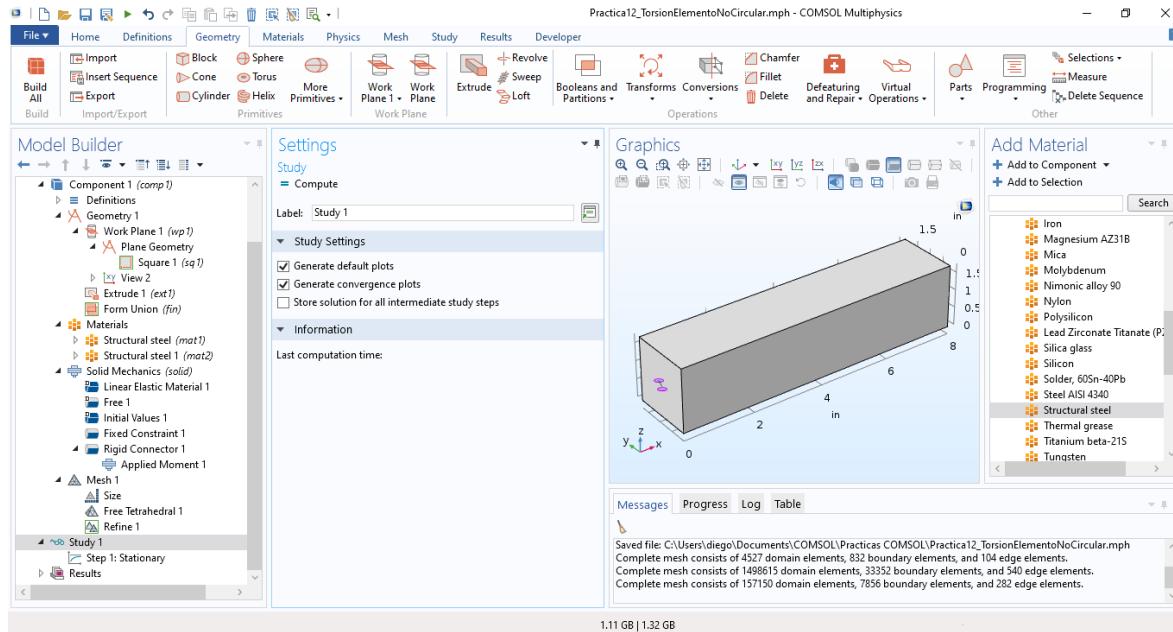


Esto se realiza para que todos los puntos de una cara reciban fuerzas de tensión, compresión o momento (torsión).



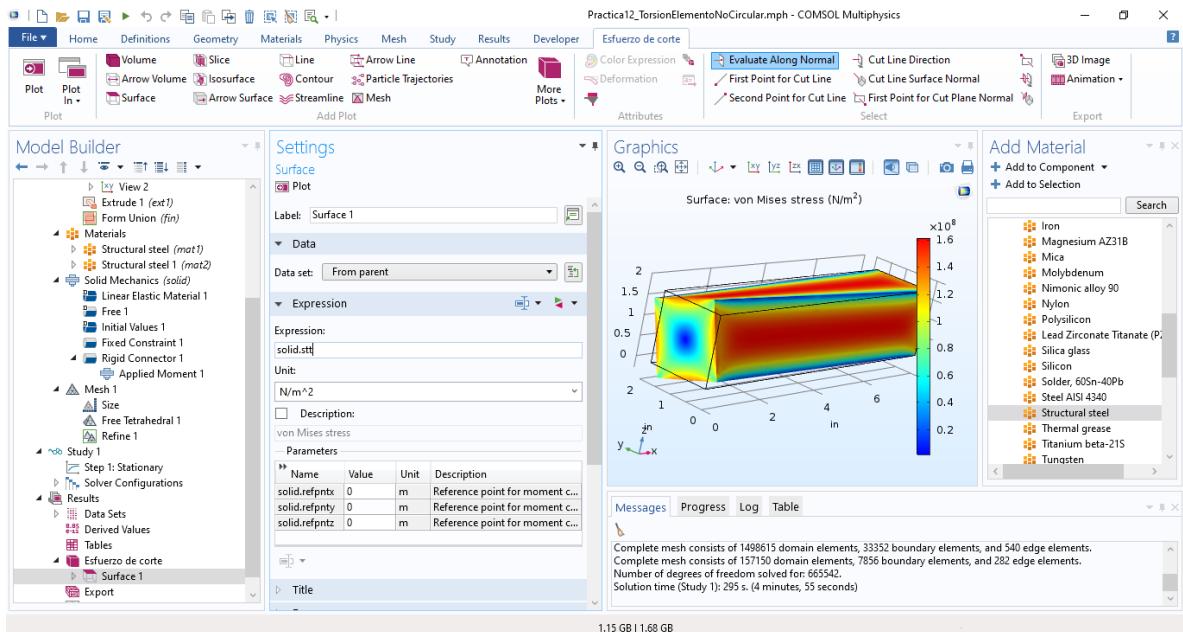


RESULTADO DEL ELEMENTO FINITO EN COMSOL:

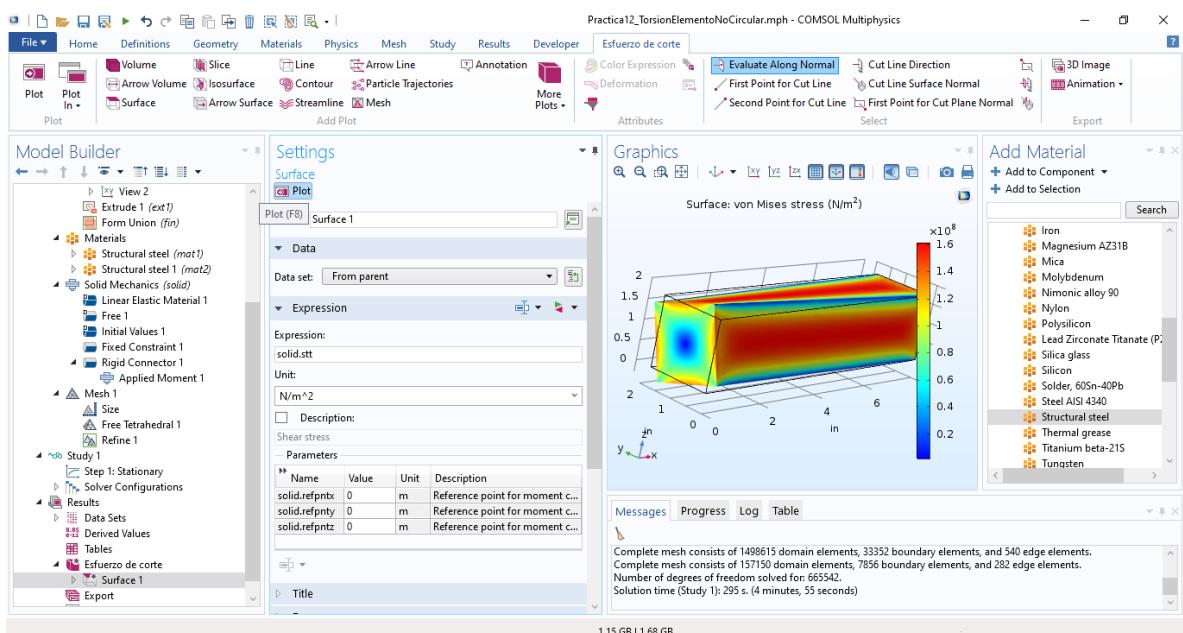


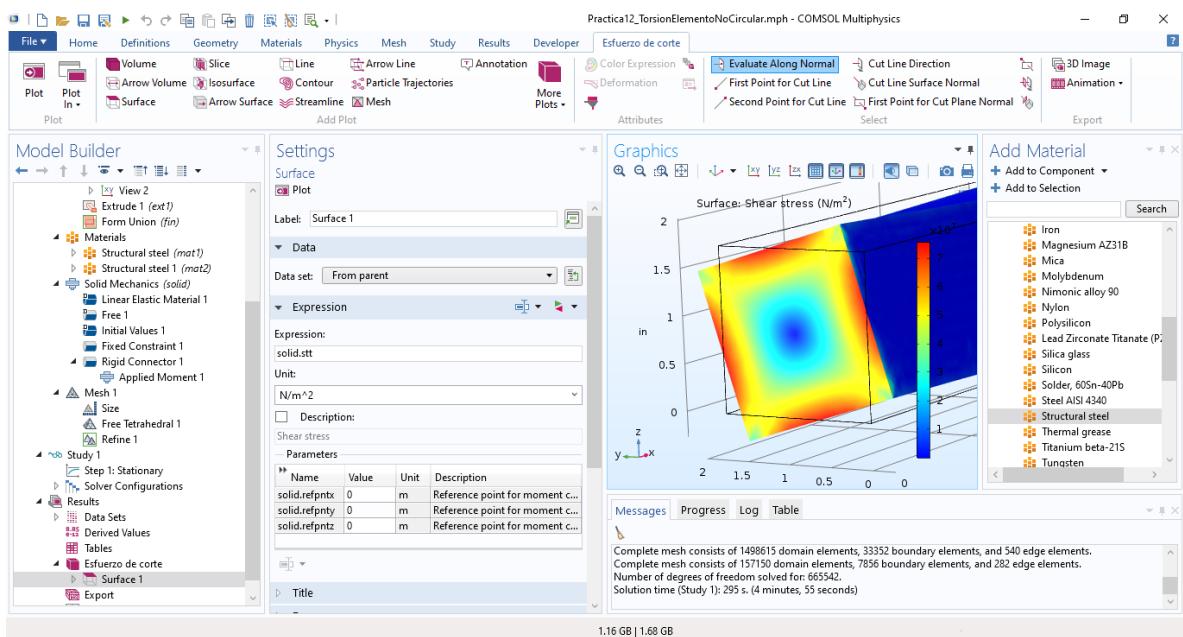
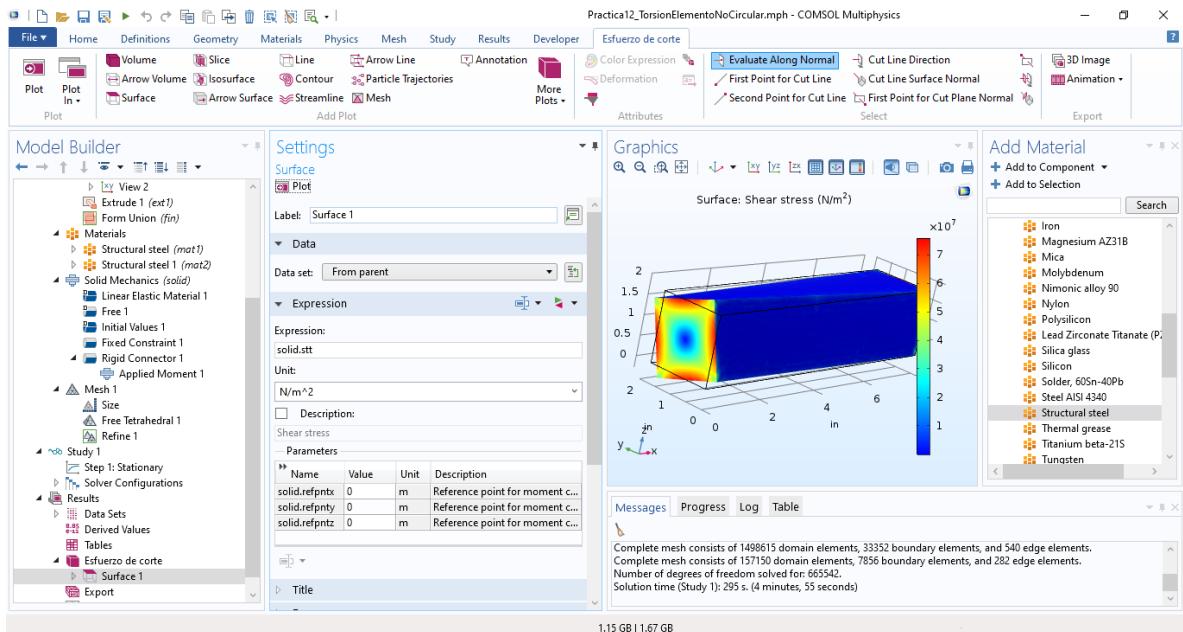
Podemos ver que el esfuerzo se concentra en las partes centrales del elemento, esto solo se puede calcular por medio de un modelo, ya que el cálculo analítico tiene la limitante de que el área de sección transversal debe ser redonda.

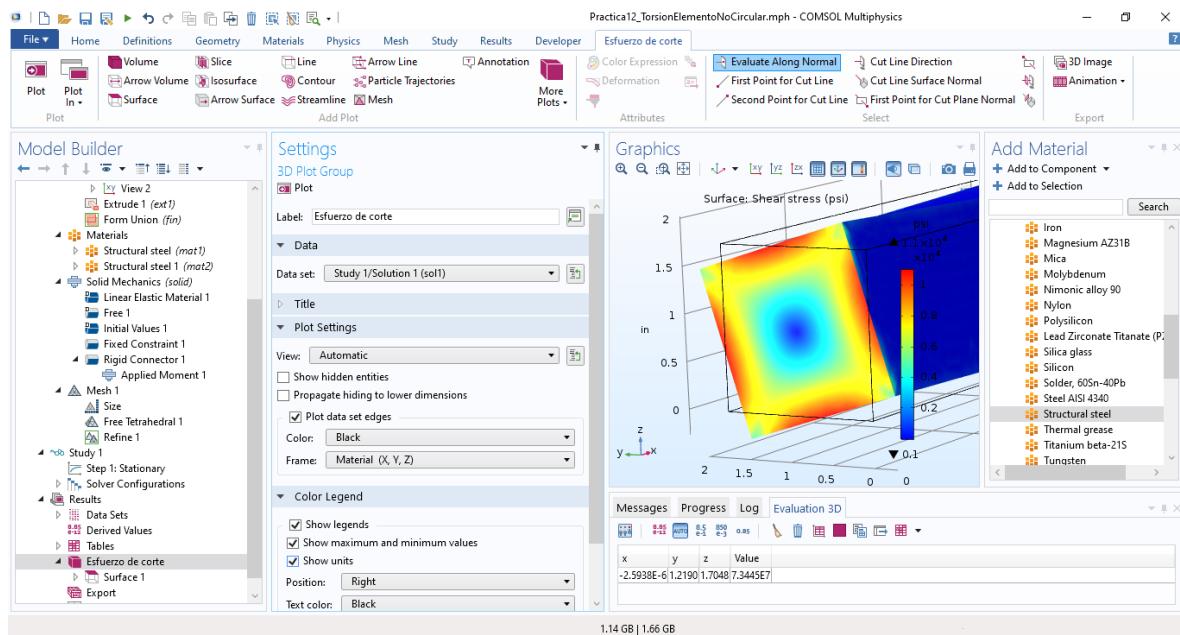
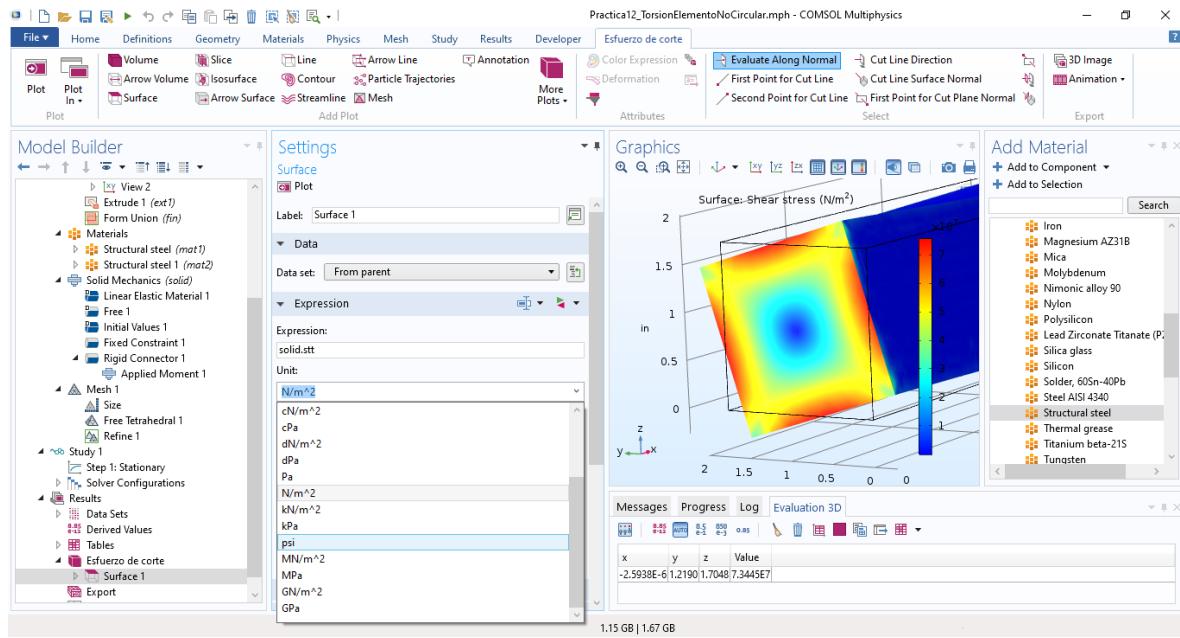


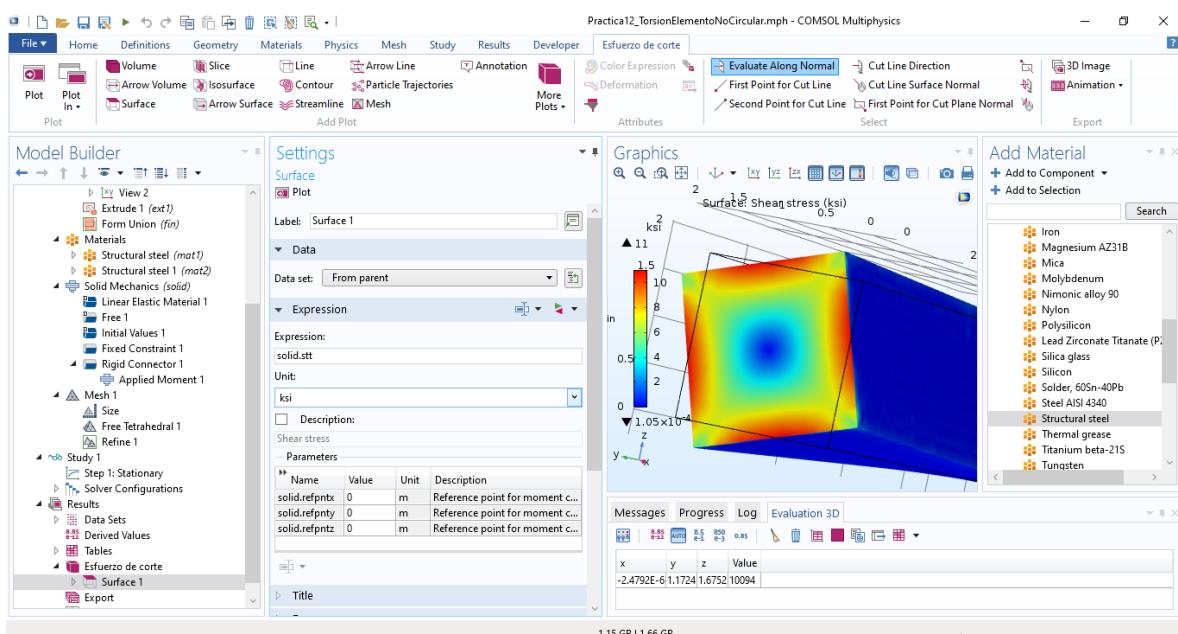
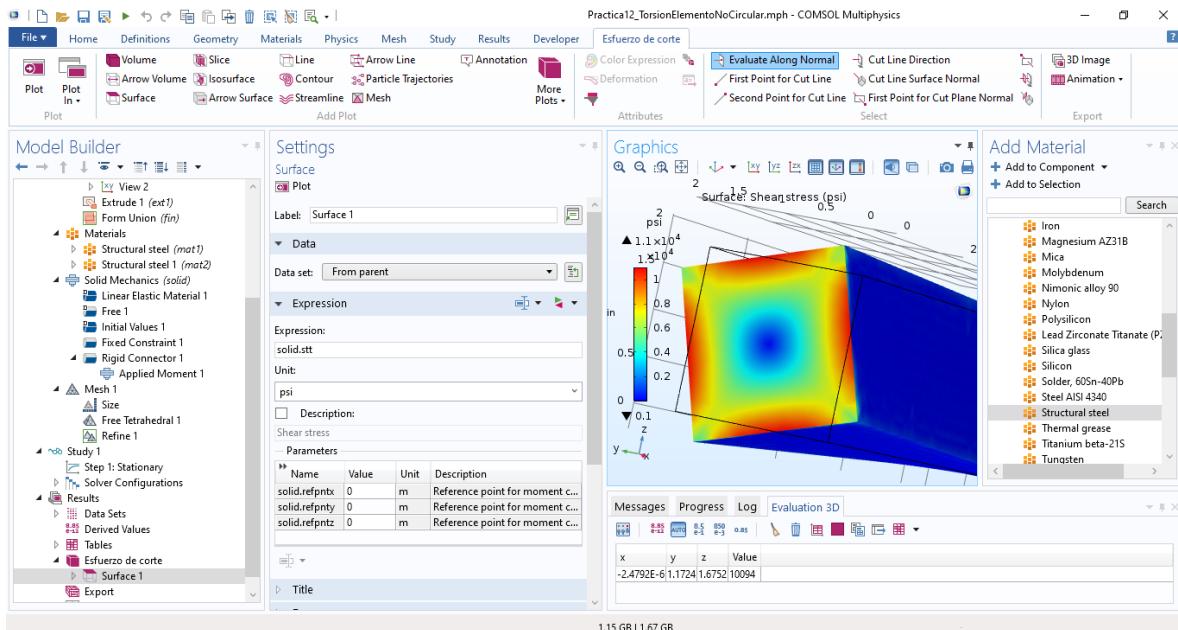


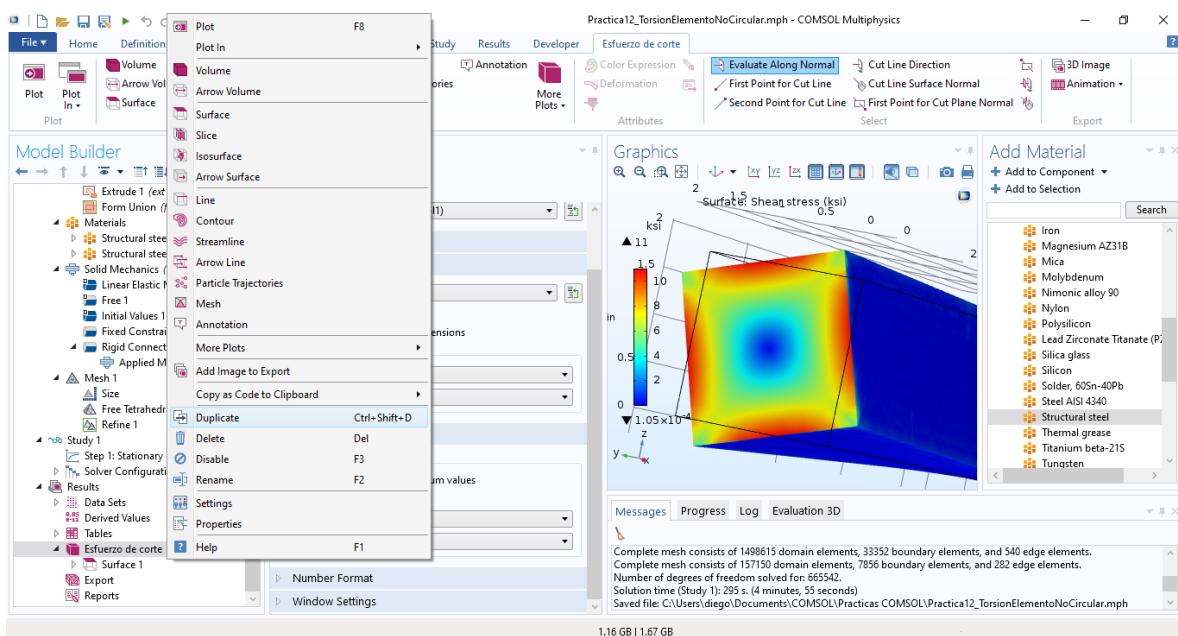
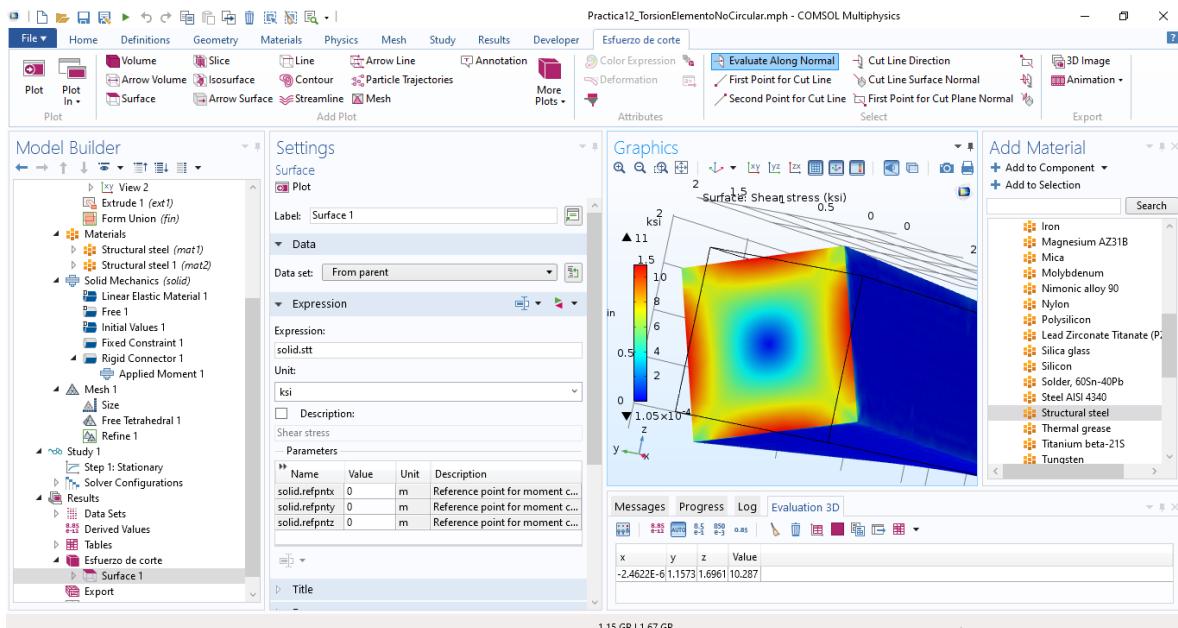
Para ver el esfuerzo de corte de la pieza se utiliza el código: solid.stt



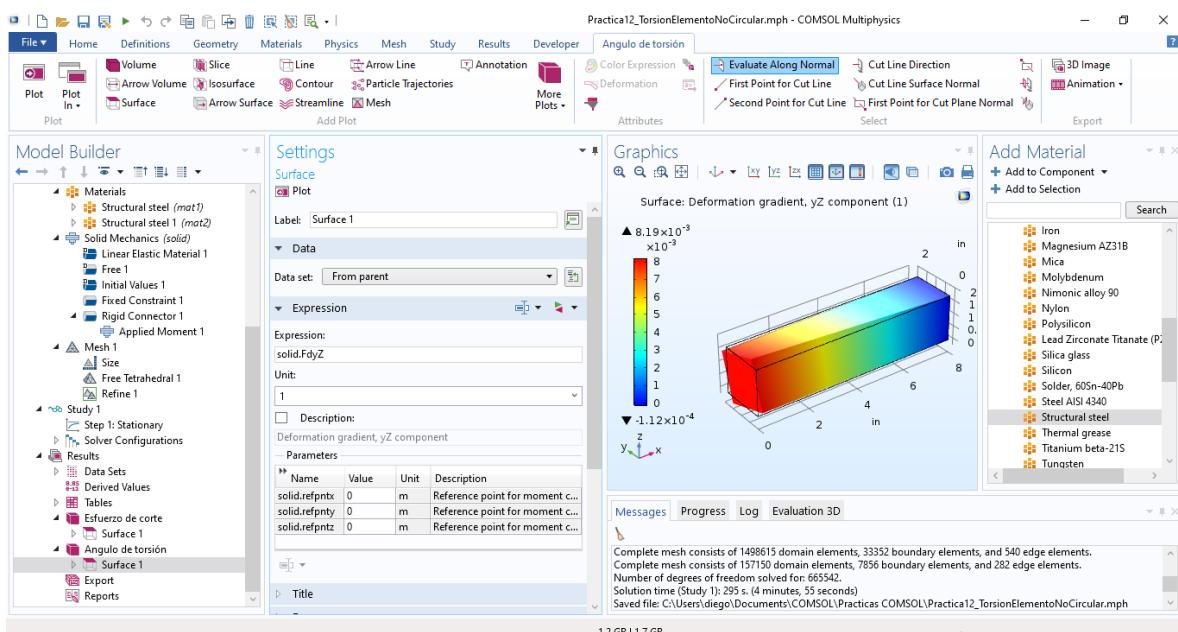
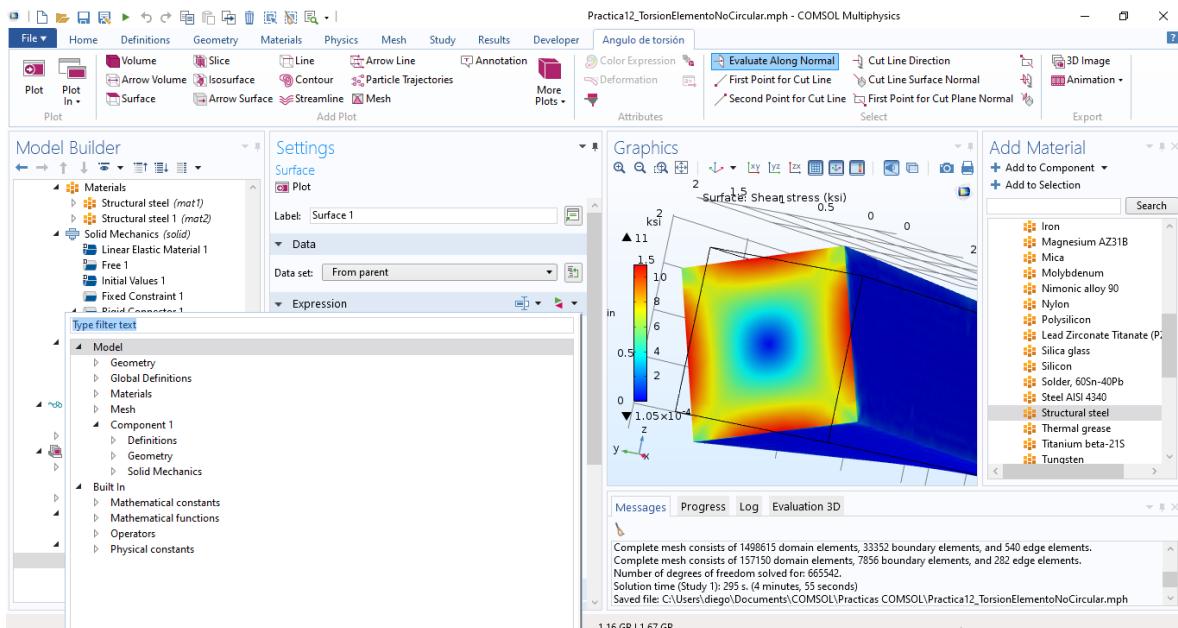




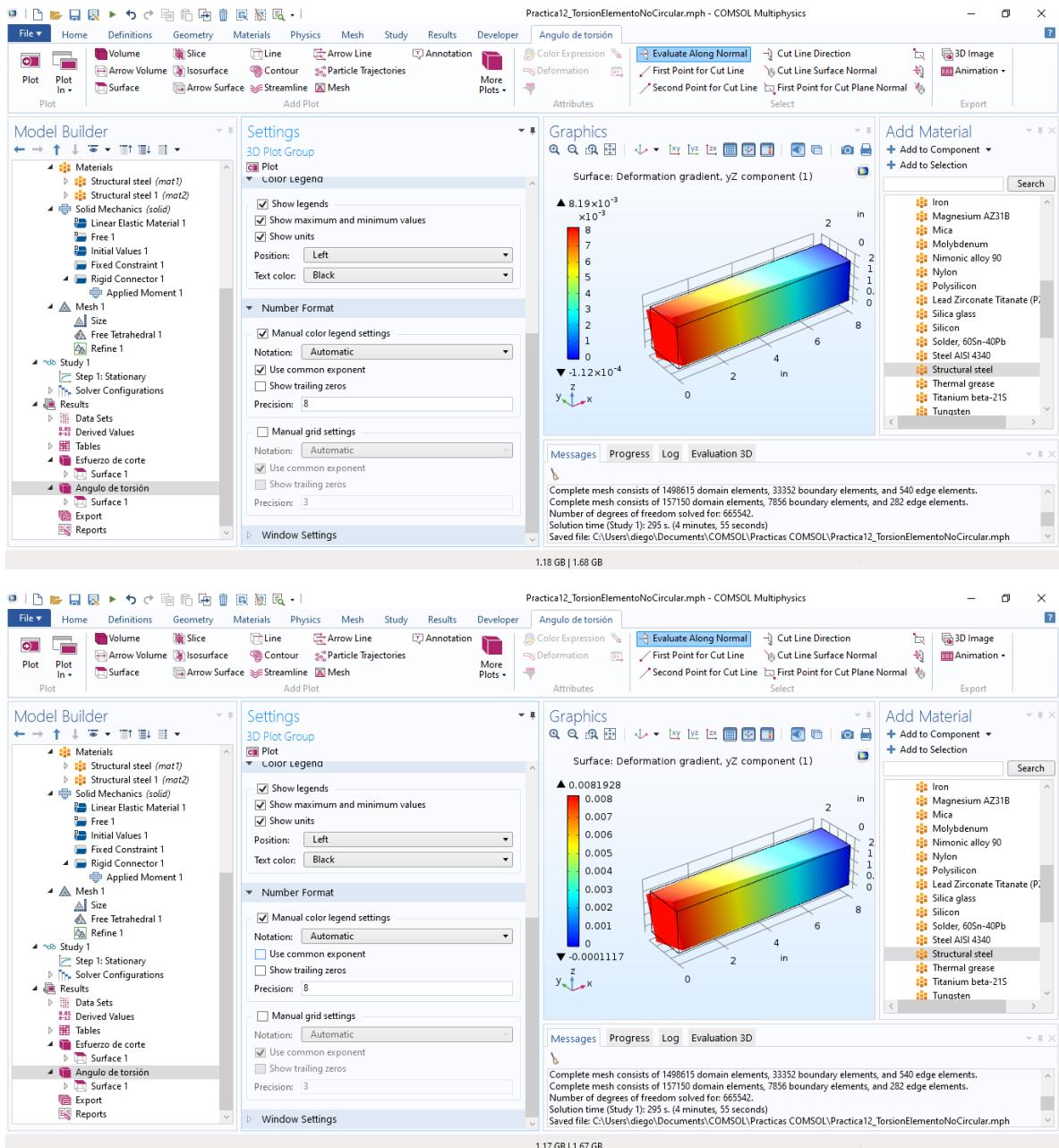


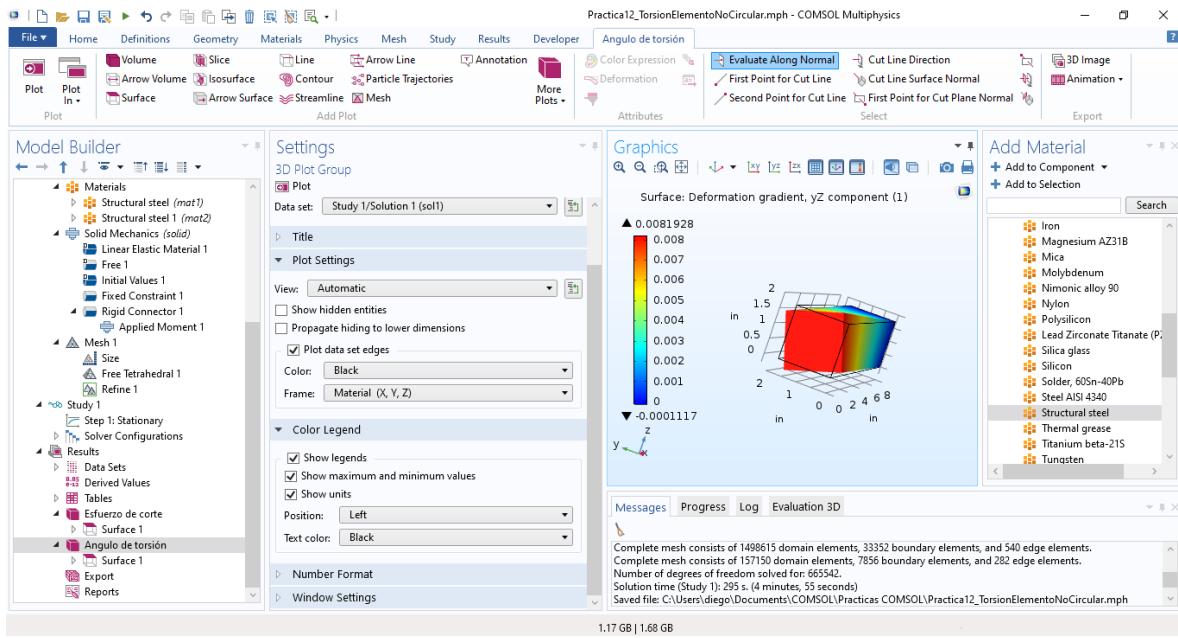


Para saber cuál es el ángulo de torsión dentro del programa COMSOL Multiphysics nos debemos introducir a la opción: Results → Esfuerzo → Surface → Expression (Flecha de hasta la derecha) → Component → Solid Mechanics → Dentro de esta opción se muestran todas las cosas que se pueden medir del modelo, desde velocidad, desplazamiento, **esfuerzo (llamado como tensión dentro del programa)**, etc.



Para encontrar el ángulo de torsión del elemento el comando que debo usar es: ***solid.FdyZ***





BIBLIOGRAFÍA:

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