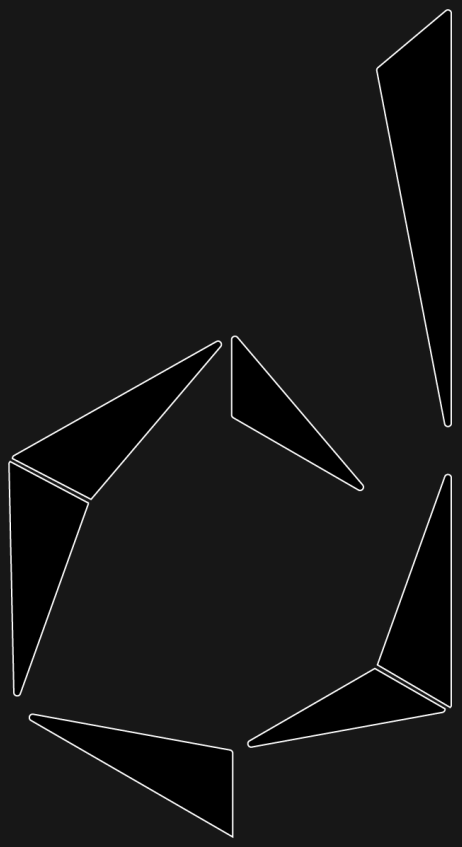
INGENIERÍA MECATRÓNICA



DI_CERO

DIEGO CERVANTES RODRÍGUEZ

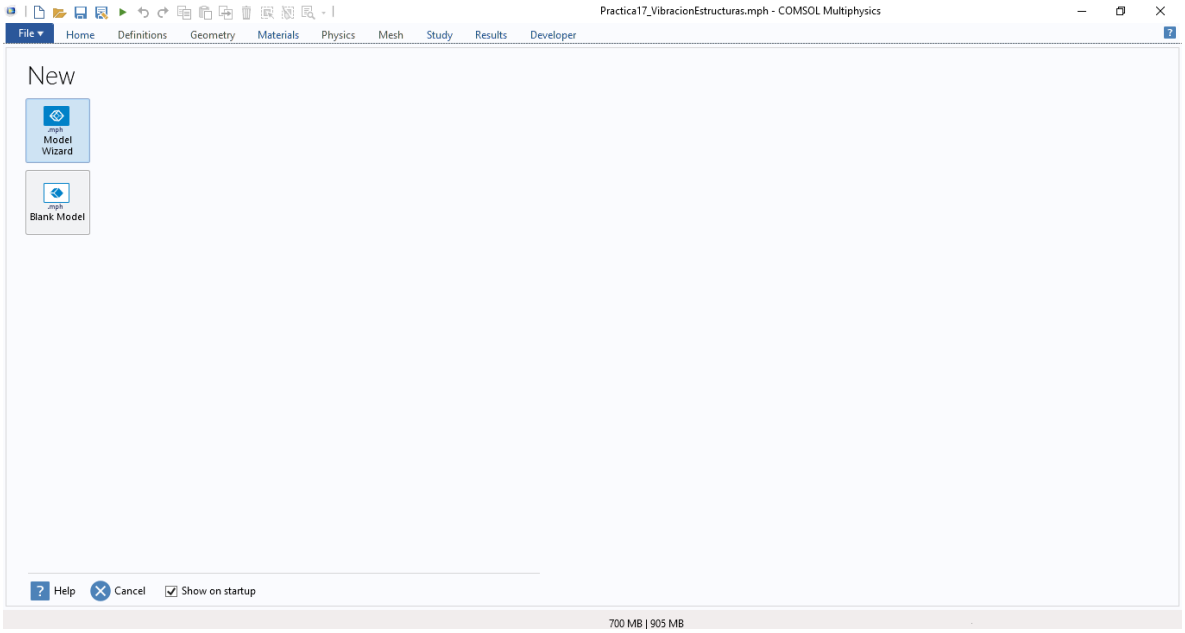
INGENIERÍA ASISTIDA POR COMPUTADORA

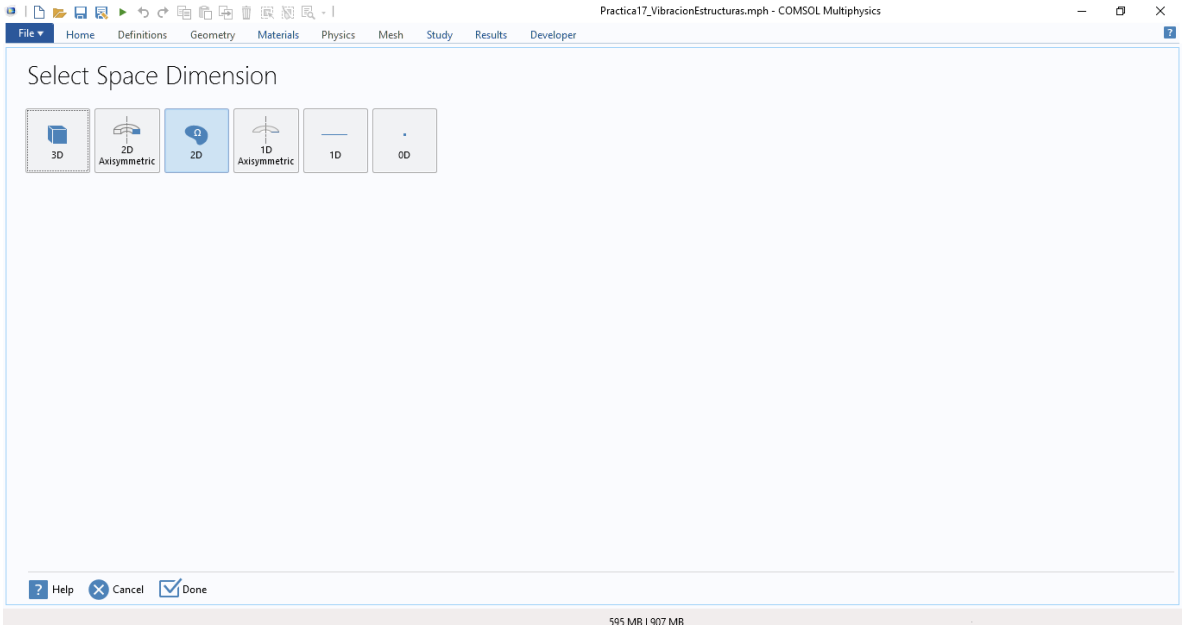
COMSOL MULTIPHYSICS 5.6

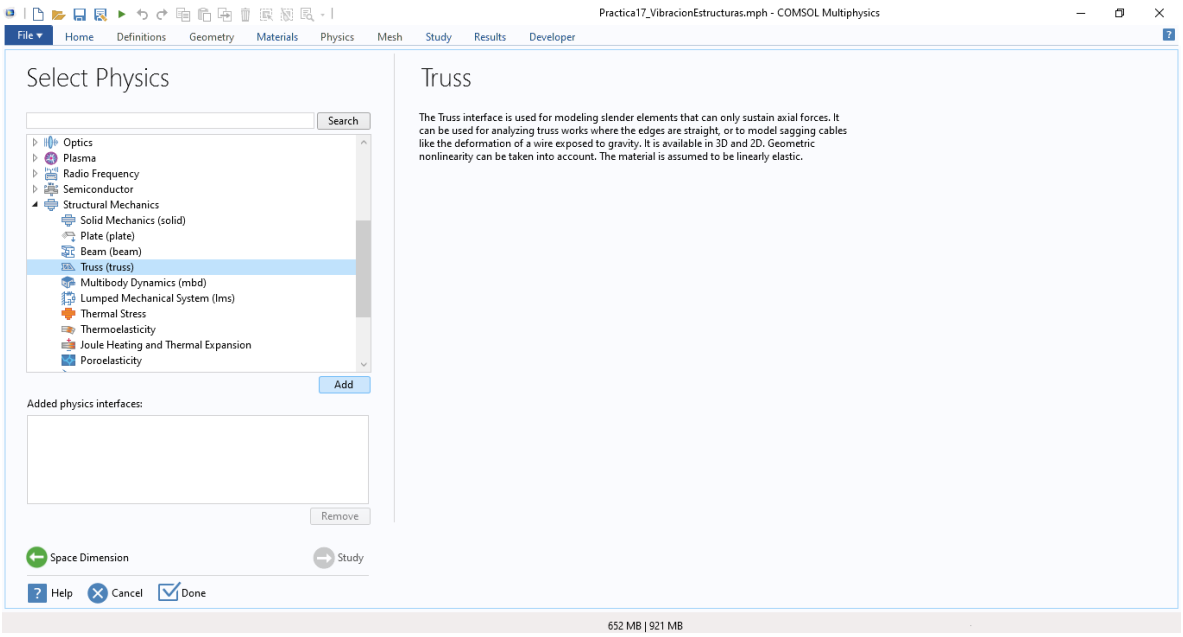
17: Vibración en Estructuras

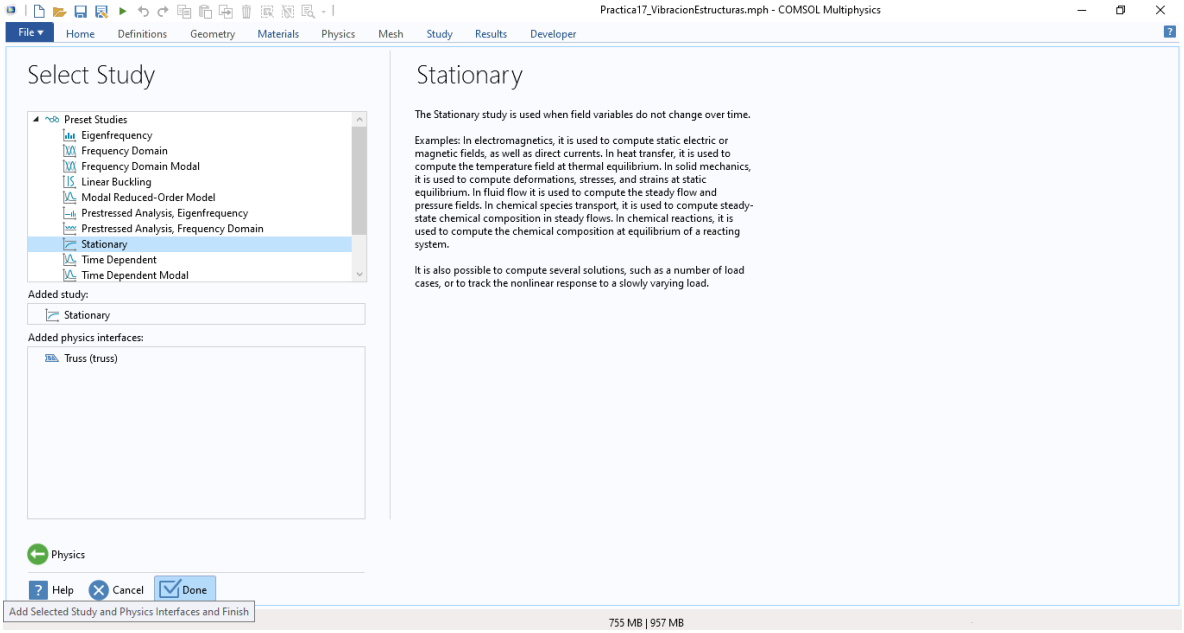
Contenido

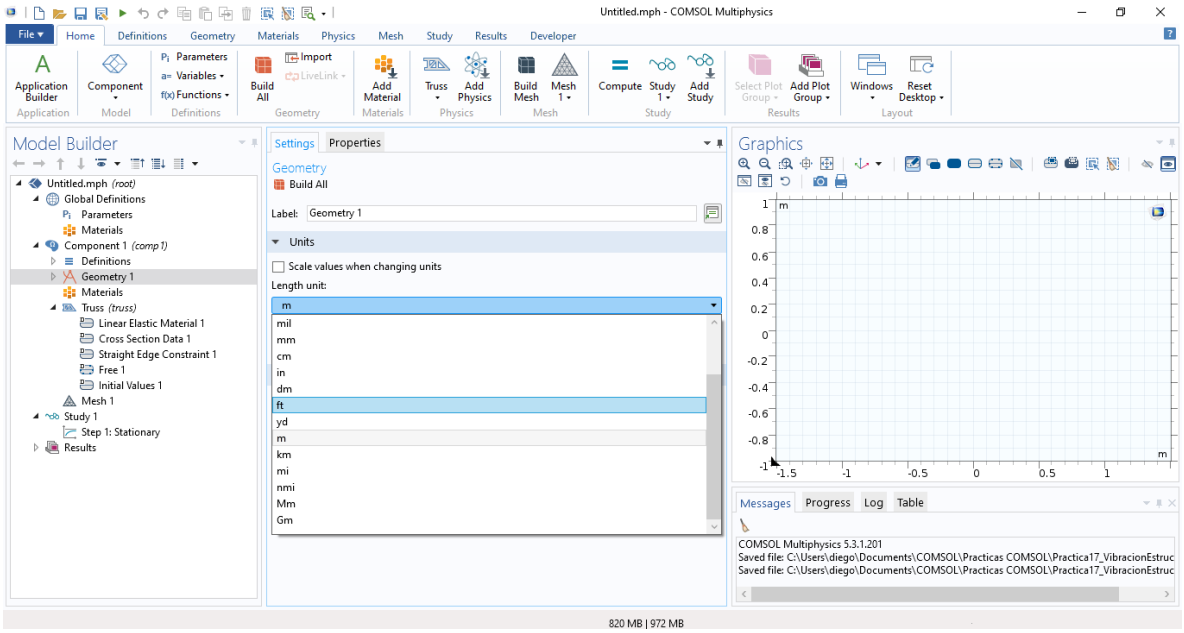
CREACIÓN DE LA PIEZA EN COMSOL:

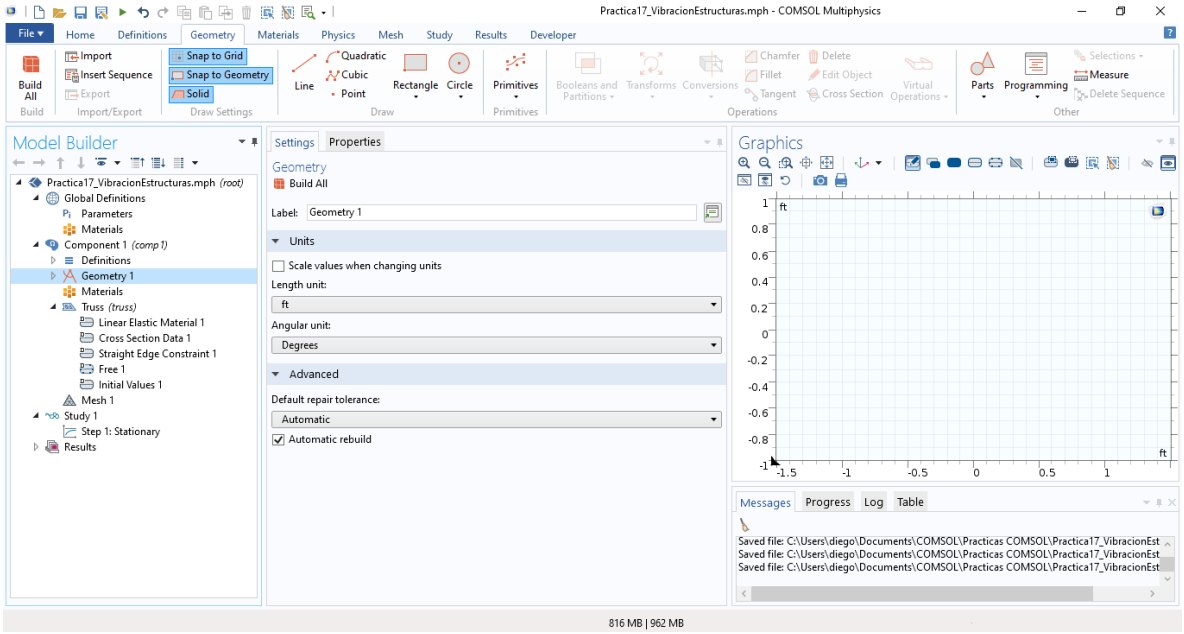


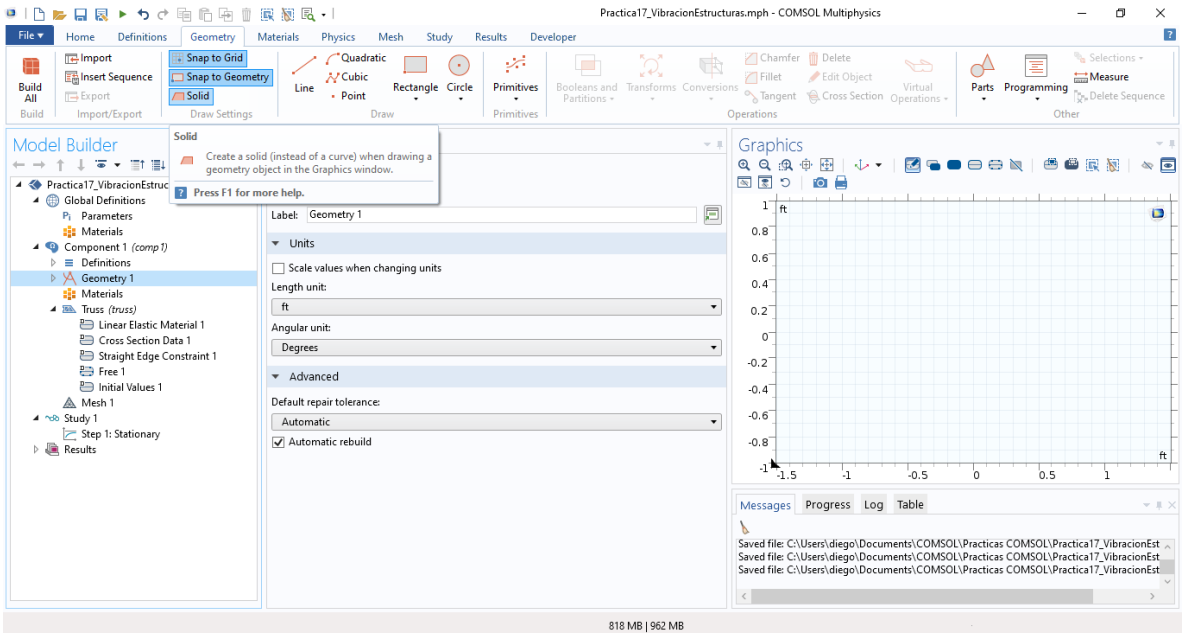


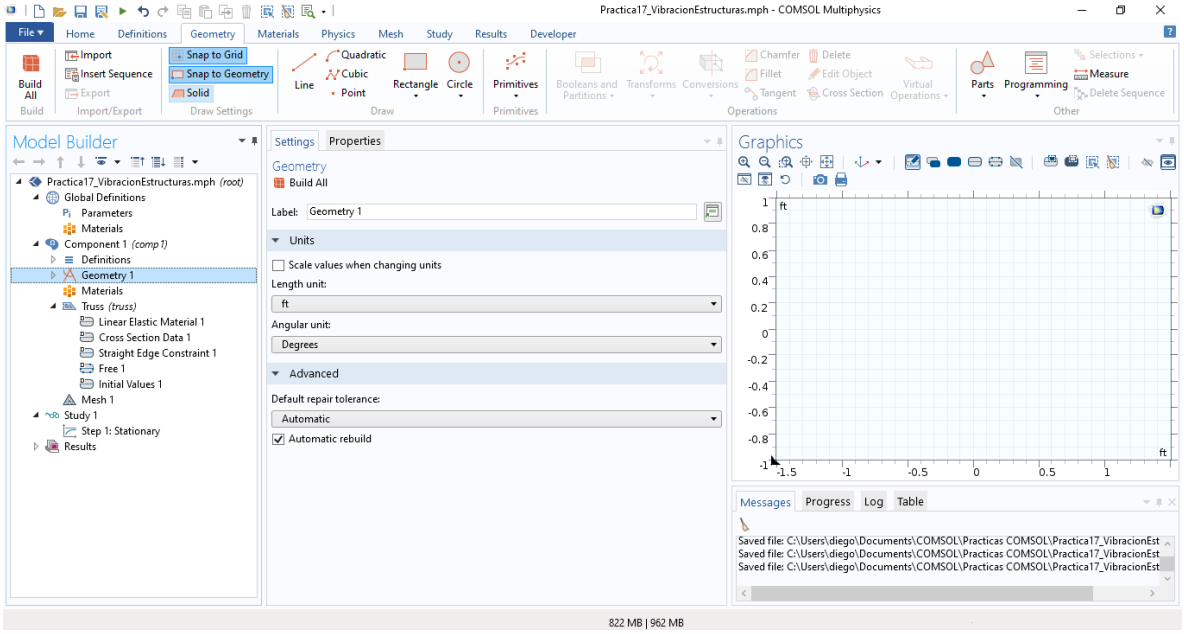

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Physics
Search
Optics
Plasma
Radio Frequency
Semiconductor
Structural Mechanics
Solid Mechanics (solid)
Plate (plate)
Beam (beam)
Truss (truss)
Multibody Dynamics (mbd)
Lumped Mechanical System (lms)
Thermal Stress
Thermoelasticity
Joule Heating and Thermal Expansion
Poroelasticity
Add
Added physics interfaces:
Remove
Space Dimension
Study
Help
Cancel
Done
Truss
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.
652 MB | 921 MB

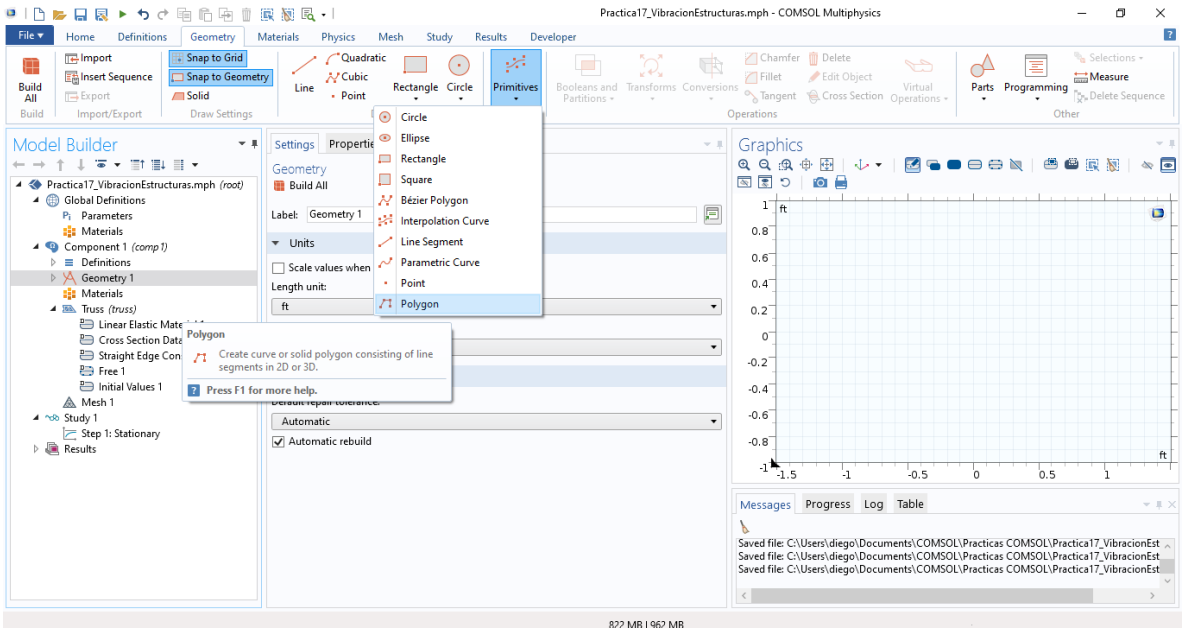

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Study
Preset Studies
Eigenfrequency
Frequency Domain
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Added study:
Stationary
Added physics interfaces:
Truss (truss)
Physics
Help
Cancel
Done
Add Selected Study and Physics Interfaces and Finish
Stationary
The Stationary study is used when field variables do not change over time.
Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute steady-state chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a reacting system.
It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.
755 MB | 957 MB

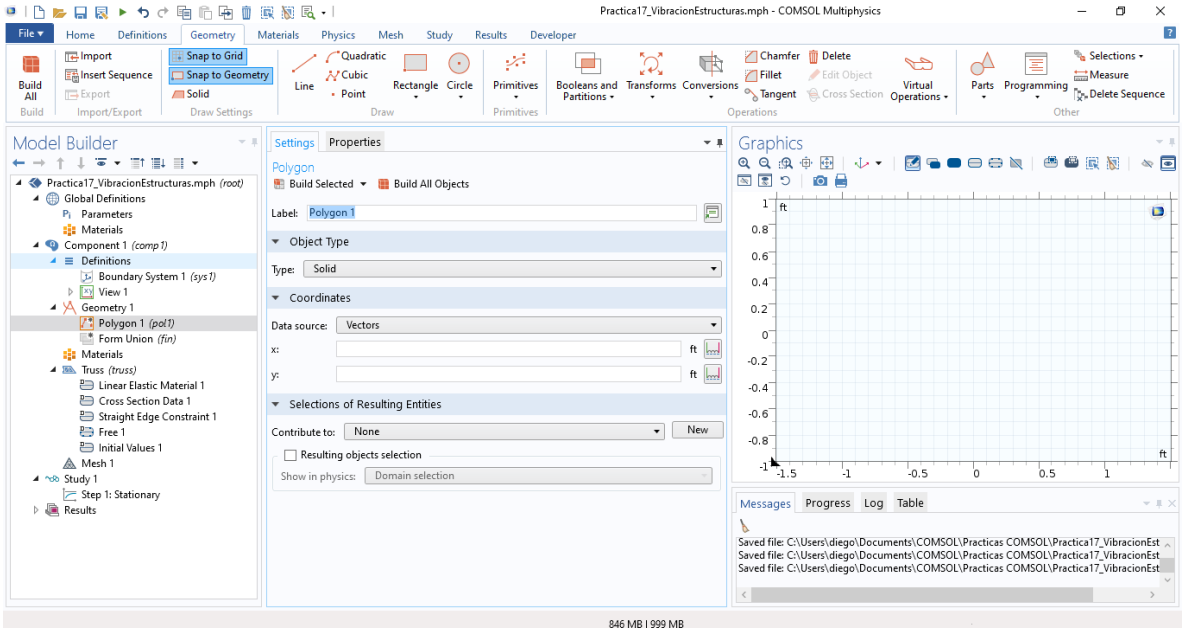

Untitled.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Model Builder
Untitled.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Geometry 1
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Geometry
Build All
Label: Geometry 1
Units
Scale values when changing units
Length unit:
m
mil
mm
cm
in
dm
ft
yd
m
km
mi
nmi
Mm
Gm
Graphics
Messages
Progress
Log
Table
COMSOL Multiphysics 5.3.1.201
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstruc
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstruc
820 MB | 972 MB

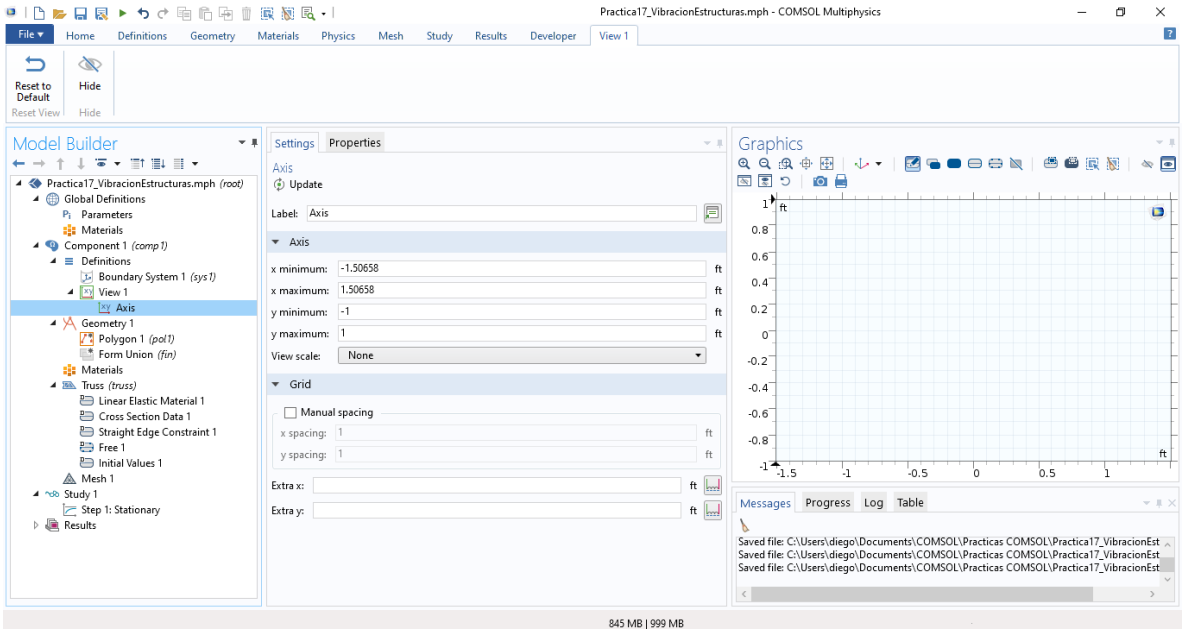

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Build All
Import
Insert Sequence
Export
Snap to Grid
Snap to Geometry
Solid
Line
Quadratic
Cubic
Point
Rectangle
Circle
Primitives
Booleans and Partitions
Transforms
Conversions
Chamfer
Fillet
Tangent
Delete
Edit Object
Cross Section
Virtual Operations
Parts
Programming
Selections
Measure
Delete Sequence
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Geometry 1
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Geometry
Build All
Label: Geometry 1
Units
Scale values when changing units
Length unit:
ft
Angular unit:
Degrees
Advanced
Default repair tolerance:
Automatic
Automatic rebuild
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
816 MB | 962 MB

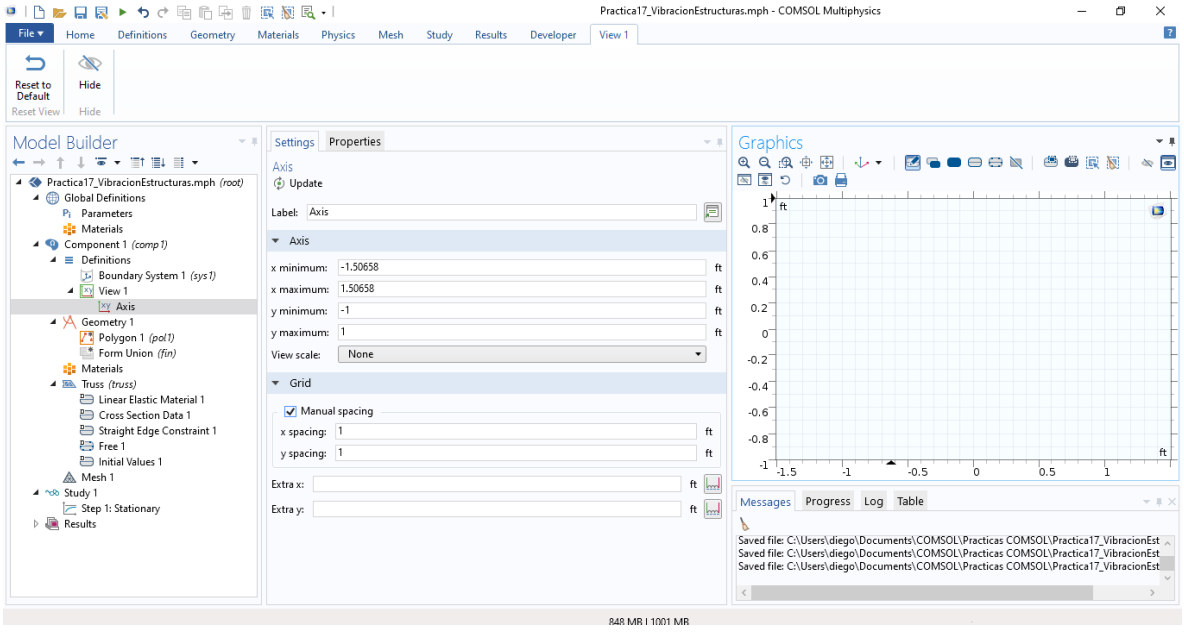

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Build All
Import
Insert Sequence
Export
Snap to Grid
Snap to Geometry
Solid
Line
Quadratic
Cubic
Point
Rectangle
Circle
Primitives
Booleans and Partitions
Transforms
Conversions
Chamfer
Fillet
Tangent
Delete
Edit Object
Cross Section
Virtual Operations
Parts
Programming
Selections
Measure
Delete Sequence
Build
Import/Export
Draw Settings
Draw
Primitives
Operations
Other
Model Builder
Solid
Create a solid (instead of a curve) when drawing a geometry object in the Graphics window.
Press F1 for more help.
Practica17_VibracionEstruc
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Geometry 1
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Label: Geometry 1
Units
Scale values when changing units
Length unit:
ft
Angular unit:
Degrees
Advanced
Default repair tolerance:
Automatic
Automatic rebuild
Graphics
ft
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
818 MB | 962 MB

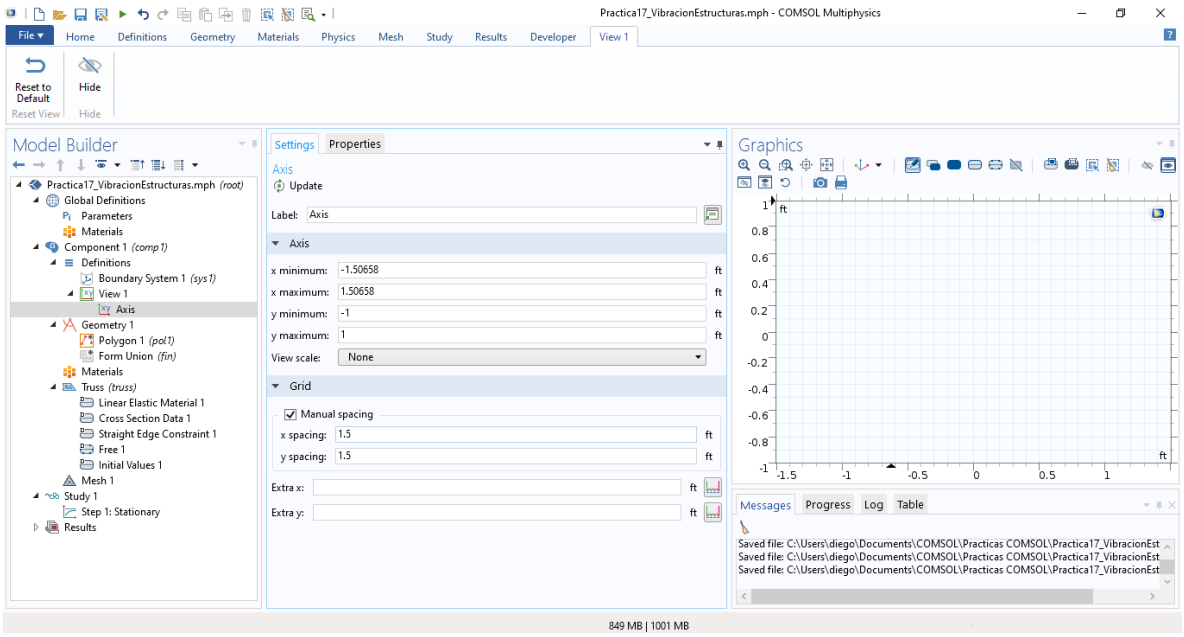

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Build All
Import
Insert Sequence
Export
Snap to Grid
Snap to Geometry
Solid
Line
Quadratic
Cubic
Point
Rectangle
Circle
Primitives
Booleans and Partitions
Transforms
Conversions
Chamfer
Fillet
Tangent
Delete
Edit Object
Cross Section
Virtual Operations
Parts
Programming
Selections
Measure
Delete Sequence
Build
Import/Export
Draw Settings
Draw
Primitives
Operations
Other
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Geometry 1
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Geometry
Build All
Label: Geometry 1
Units
Scale values when changing units
Length unit:
ft
Angular unit:
Degrees
Advanced
Default repair tolerance:
Automatic
Automatic rebuild
Graphics
ft
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
822 MB | 962 MB

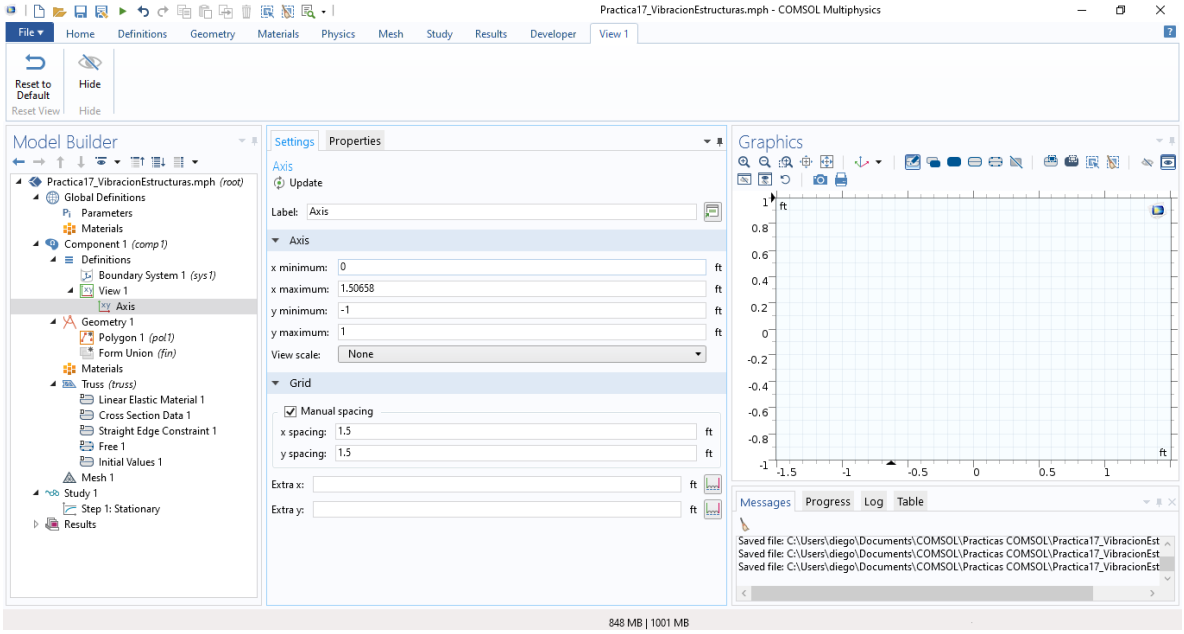

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Build All
Import
Insert Sequence
Export
Snap to Grid
Snap to Geometry
Solid
Line
Quadratic
Cubic
Point
Rectangle
Circle
Primitives
Booleans and Partitions
Transforms
Conversions
Chamfer
Fillet
Tangent
Delete
Edit Object
Cross Section
Virtual Operations
Parts
Programming
Selections
Measure
Delete Sequence
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Geometry 1
Materials
Truss (truss)
Linear Elastic Mate
Cross Section Data
Straight Edge Con
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Geometry
Build All
Label: Geometry 1
Units
Scale values when
Length unit:
ft
Automatic
Automatic rebuild
Circle
Ellipse
Rectangle
Square
Bézier Polygon
Interpolation Curve
Line Segment
Parametric Curve
Point
Polygon
Polygon
Create curve or solid polygon consisting of line segments in 2D or 3D.
Press F1 for more help.
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
822 MB | 962 MB

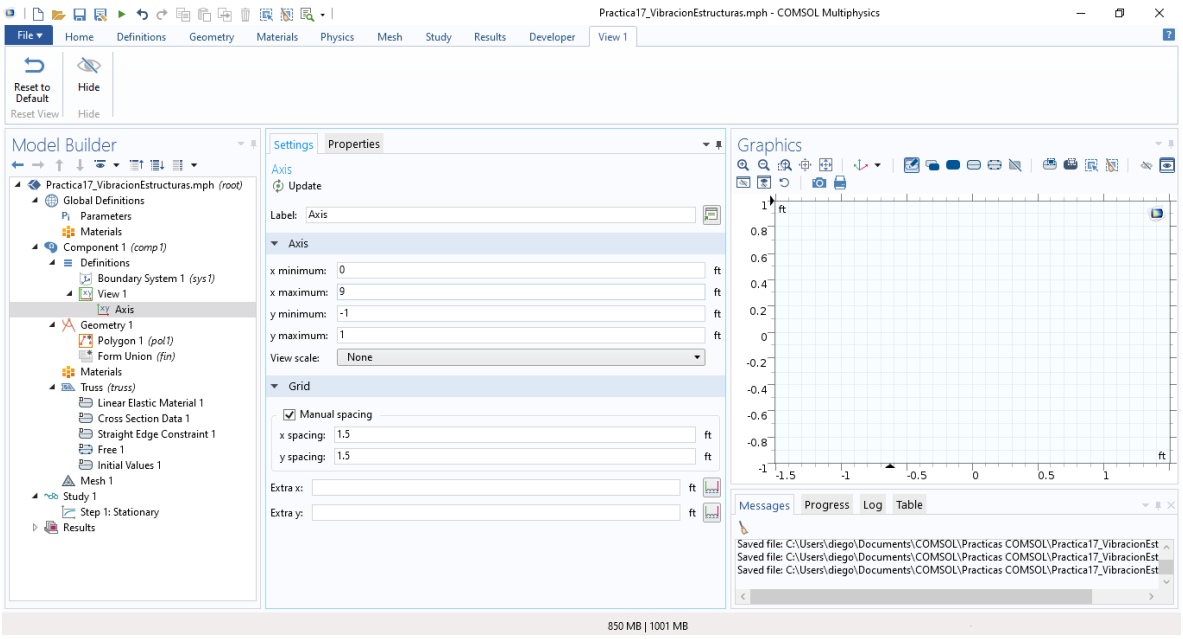

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Build All
Import
Insert Sequence
Export
Snap to Grid
Snap to Geometry
Solid
Line
Quadratic
Cubic
Point
Rectangle
Circle
Primitives
Booleans and Partitions
Transforms
Conversions
Chamfer
Fillet
Tangent
Delete
Edit Object
Cross Section
Virtual Operations
Parts
Programming
Selections
Measure
Delete Sequence
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Polygon
Build Selected
Build All Objects
Label: Polygon 1
Object Type
Type: Solid
Coordinates
Data source: Vectors
x:
y:
ft
Selections of Resulting Entities
Contribute to: None
New
Resulting objects selection
Show in physics: Domain selection
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
846 MB | 999 MB

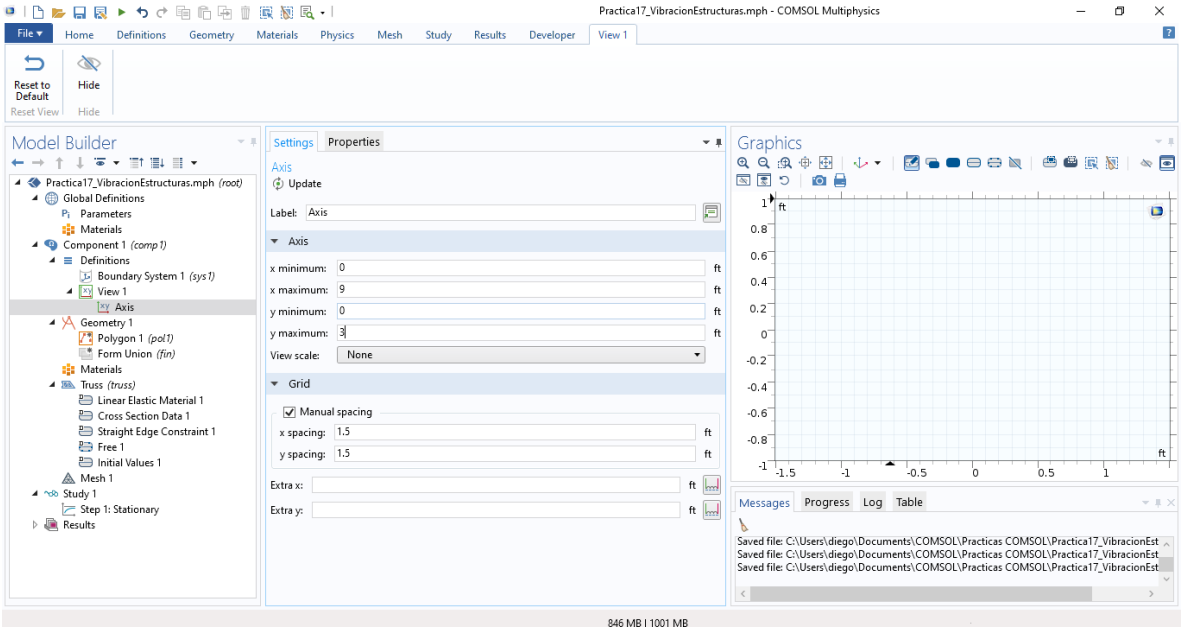

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: -1.50658
x maximum: 1.50658
y minimum: -1
y maximum: 1
View scale: None
Grid
Manual spacing
x spacing: 1
y spacing: 1
Extra x:
Extra y:
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
845 MB | 999 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: -1.50658
x maximum: 1.50658
y minimum: -1
y maximum: 1
View scale: None
Grid
Manual spacing
x spacing: 1
y spacing: 1
Extra x:
Extra y:
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
848 MB | 1001 MB

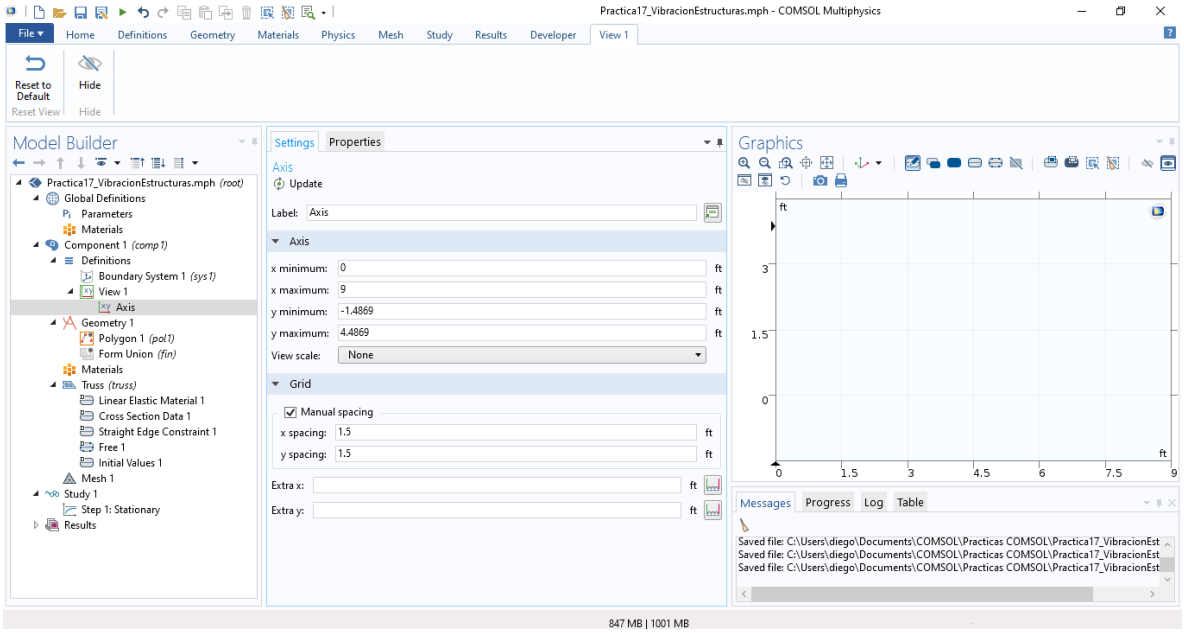

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: -1.50658
x maximum: 1.50658
y minimum: -1
y maximum: 1
View scale: None
Grid
Manual spacing
x spacing: 1.5
y spacing: 1.5
Extra x:
Extra y:
ft
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
849 MB | 1001 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: 0
x maximum: 1.50658
y minimum: -1
y maximum: 1
View scale: None
Grid
Manual spacing
x spacing: 1.5
y spacing: 1.5
Extra x:
Extra y:
ft
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
848 MB | 1001 MB

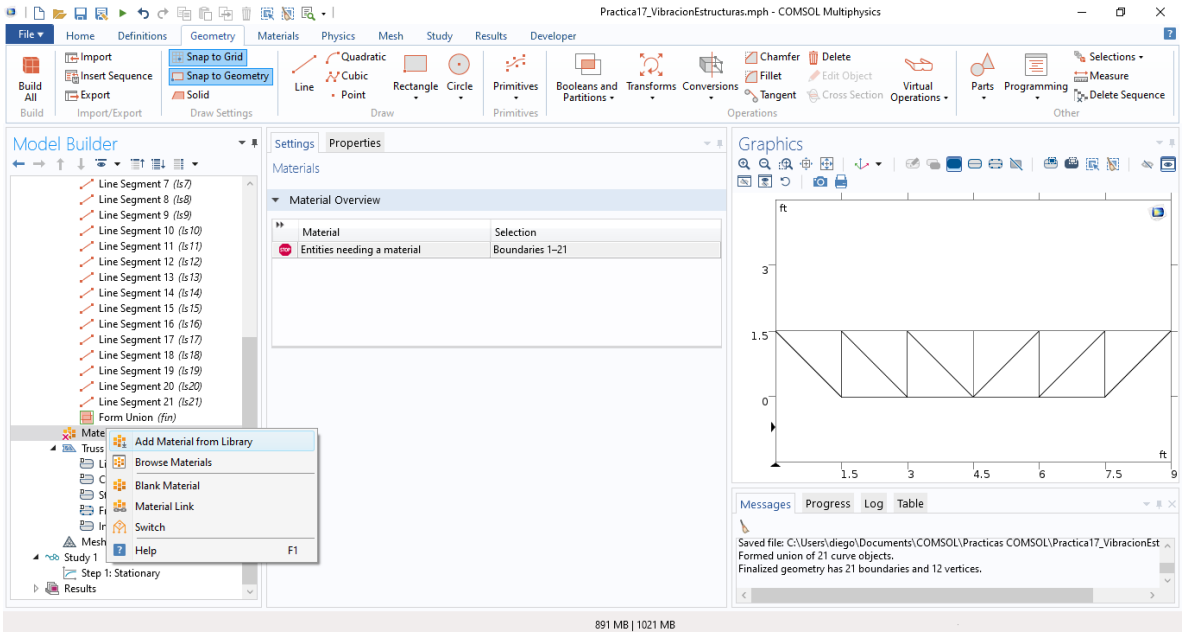

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: 0
x maximum: 9
y minimum: -1
y maximum: 1
View scale: None
Grid
Manual spacing
x spacing: 1.5
y spacing: 1.5
Extra x:
Extra y:
ft
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
850 MB | 1001 MB

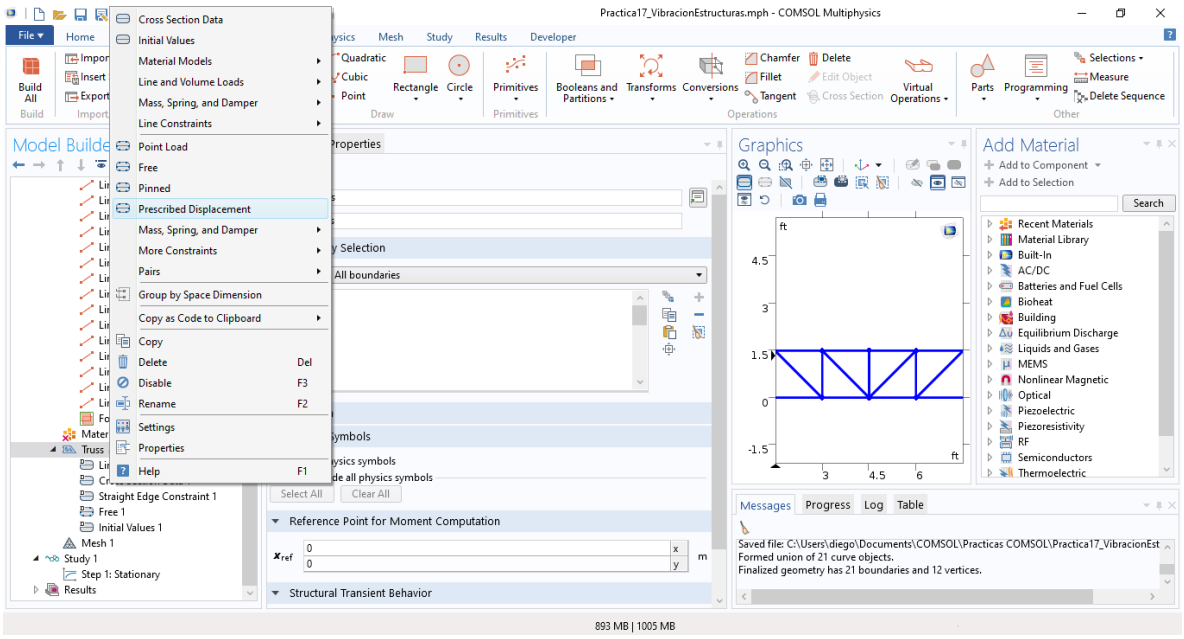

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
View 1
Reset to Default
Hide
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp 1)
Definitions
Boundary System 1 (sys1)
View 1
Axis
Geometry 1
Polygon 1 (pol1)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Cross Section Data 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Axis
Update
Label: Axis
Axis
x minimum: 0
x maximum: 9
y minimum: 0
y maximum: 3
View scale: None
Grid
Manual spacing
x spacing: 1.5
y spacing: 1.5
Extra x:
Extra y:
ft
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEst
846 MB | 1001 MB

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

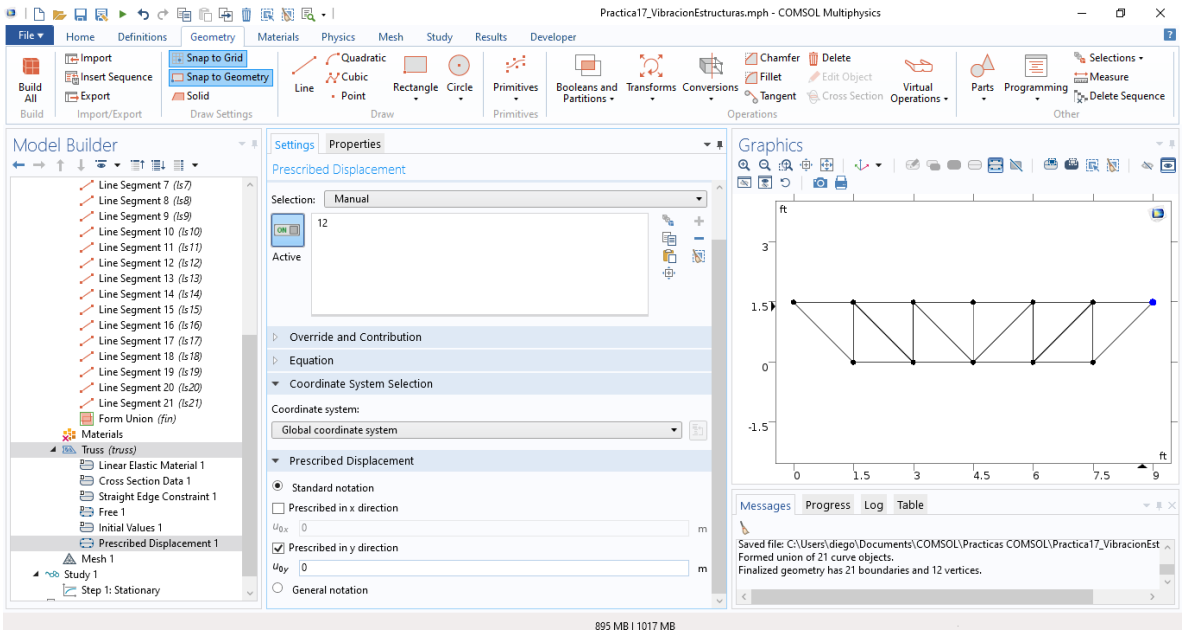


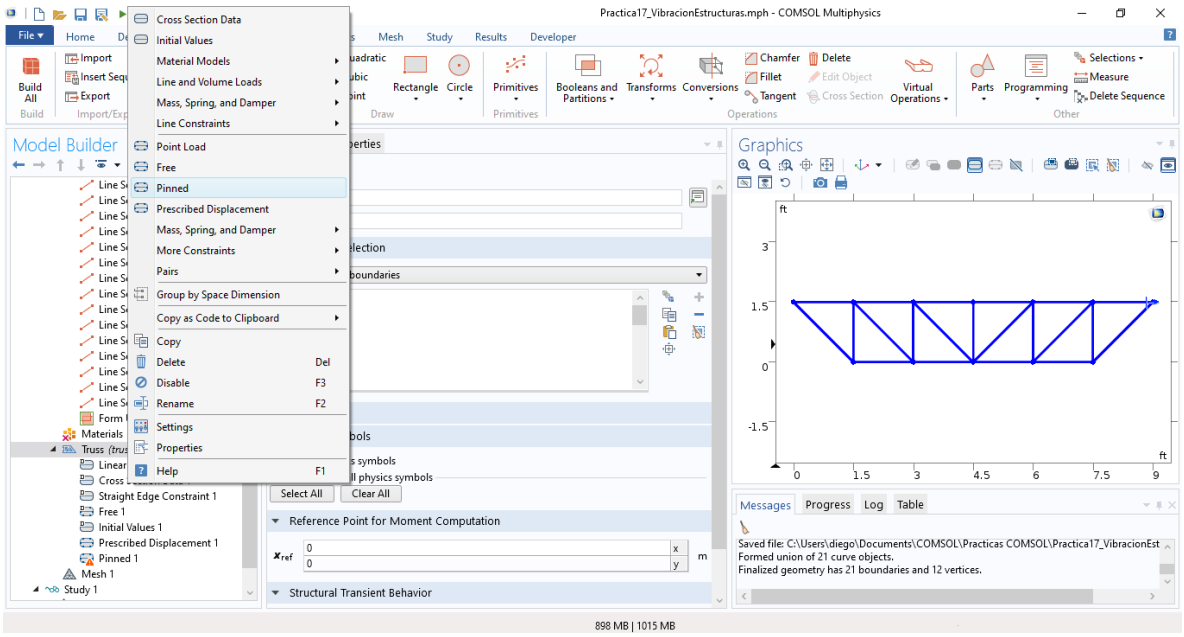
ANÁLISIS MECÁNICO EN COMSOL:

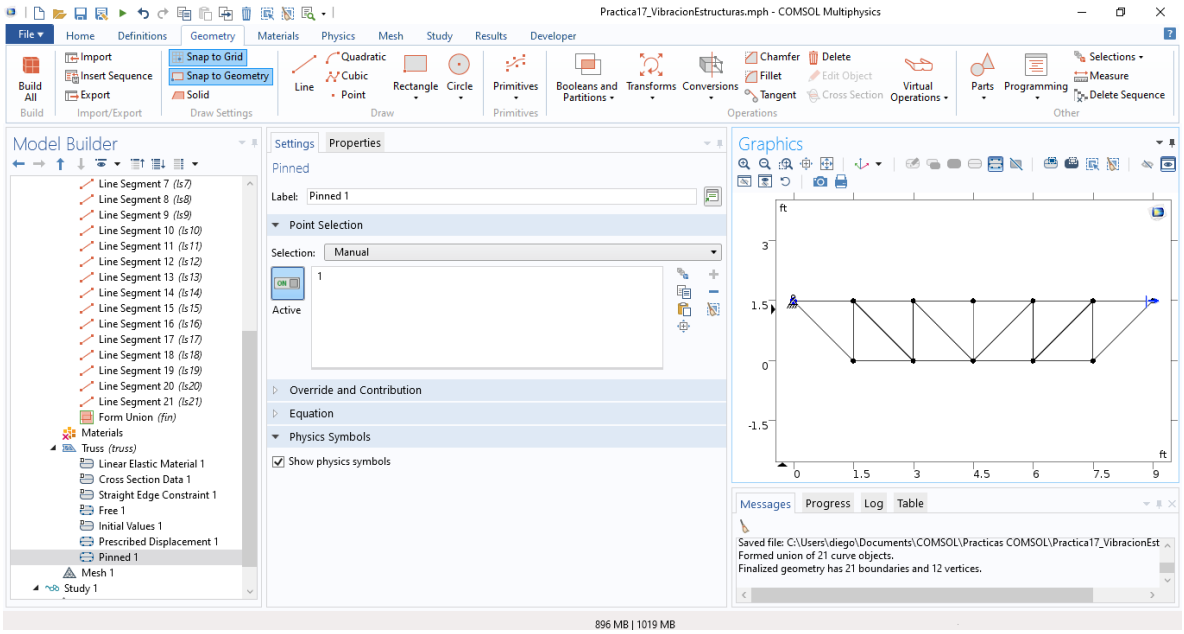


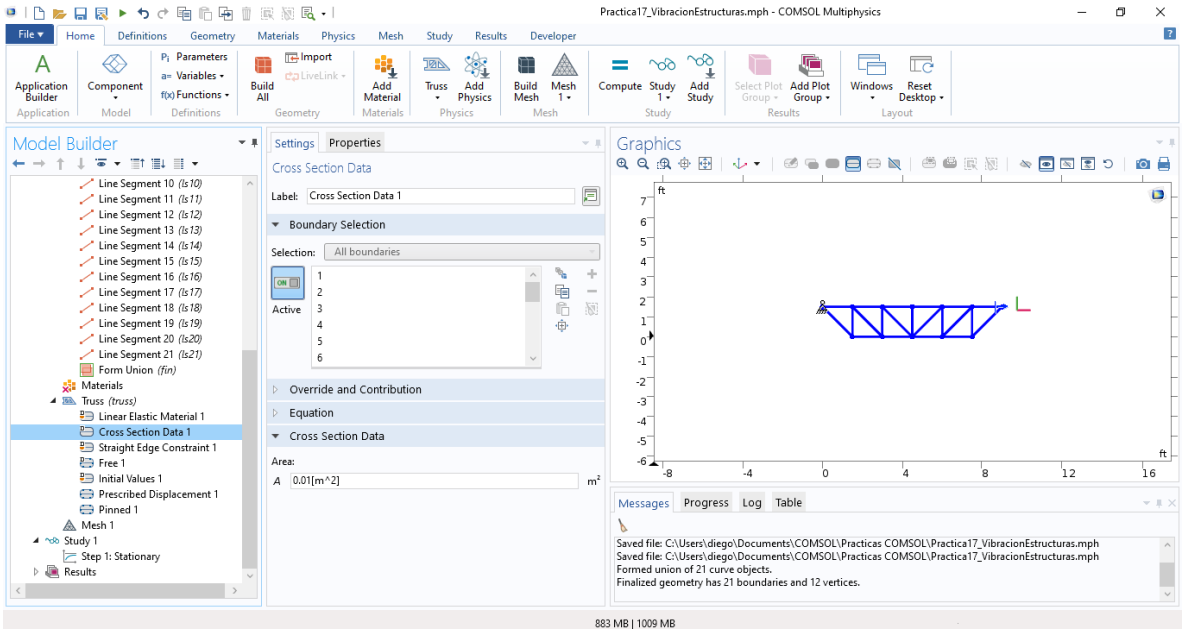


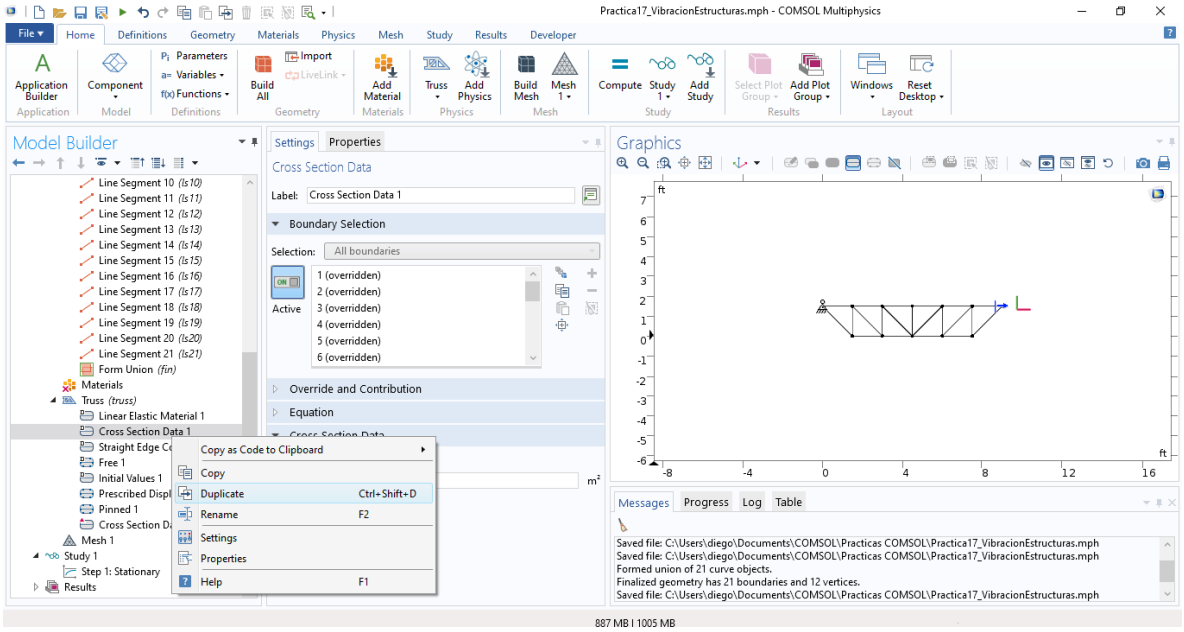
- **Prescribed Displacement:** Este apoyo no deja que se mueva verticalmente la armadura.

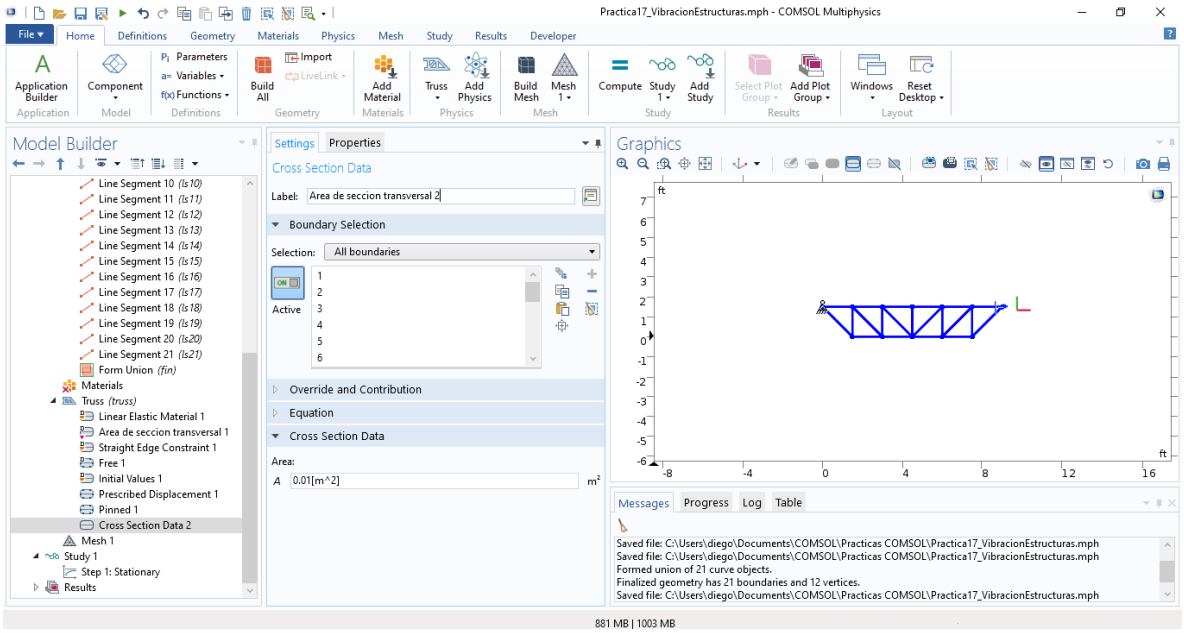


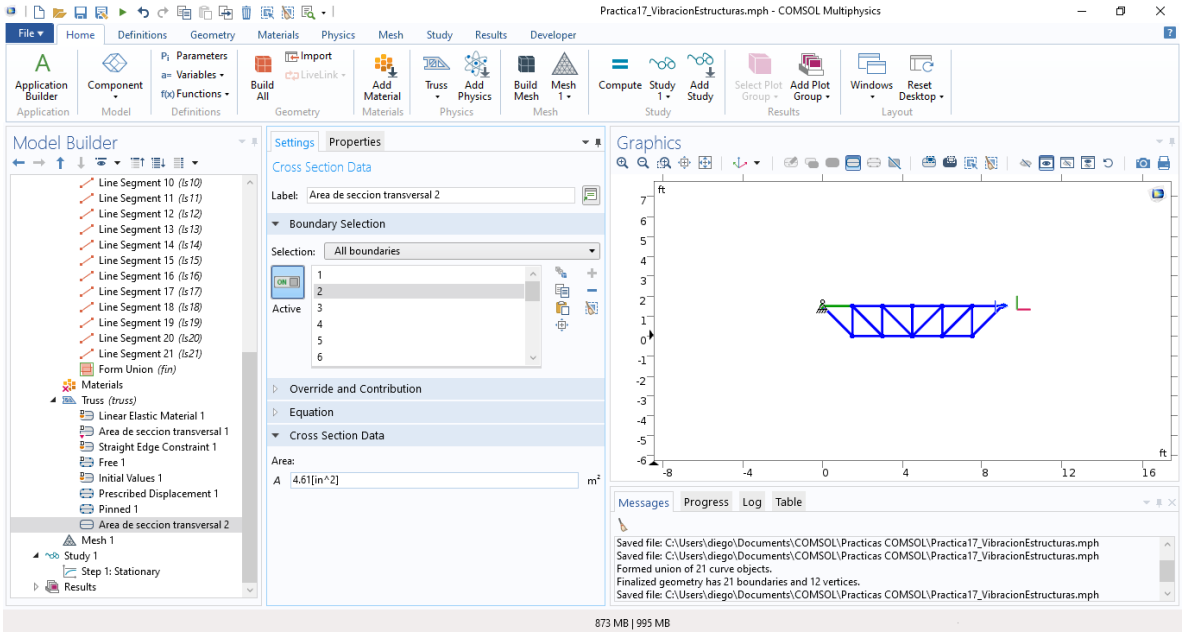

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Cross Section Data
Initial Values
Material Models
Line and Volume Loads
Mass, Spring, and Damper
Line Constraints
Point Load
Free
Pinned
Prescribed Displacement
Mass, Spring, and Damper
More Constraints
Pairs
Group by Space Dimension
Copy as Code to Clipboard
Copy
Delete Del
Disable F3
Rename F2
Settings
Properties
Help F1
Model Builder
Graphics
Reference Point for Moment Computation
Structural Transient Behavior
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
898 MB | 1015 MB

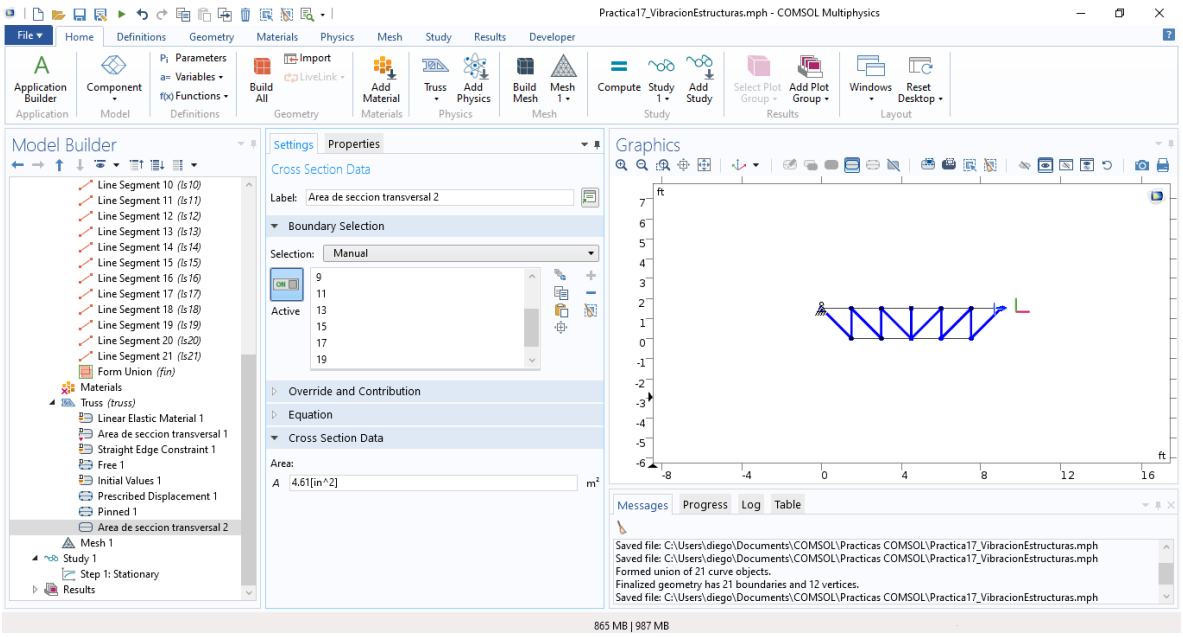

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Pinned
Label: Pinned 1
Point Selection
Selection: Manual
Active
Override and Contribution
Equation
Physics Symbols
Show physics symbols
Graphics
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
896 MB | 1019 MB

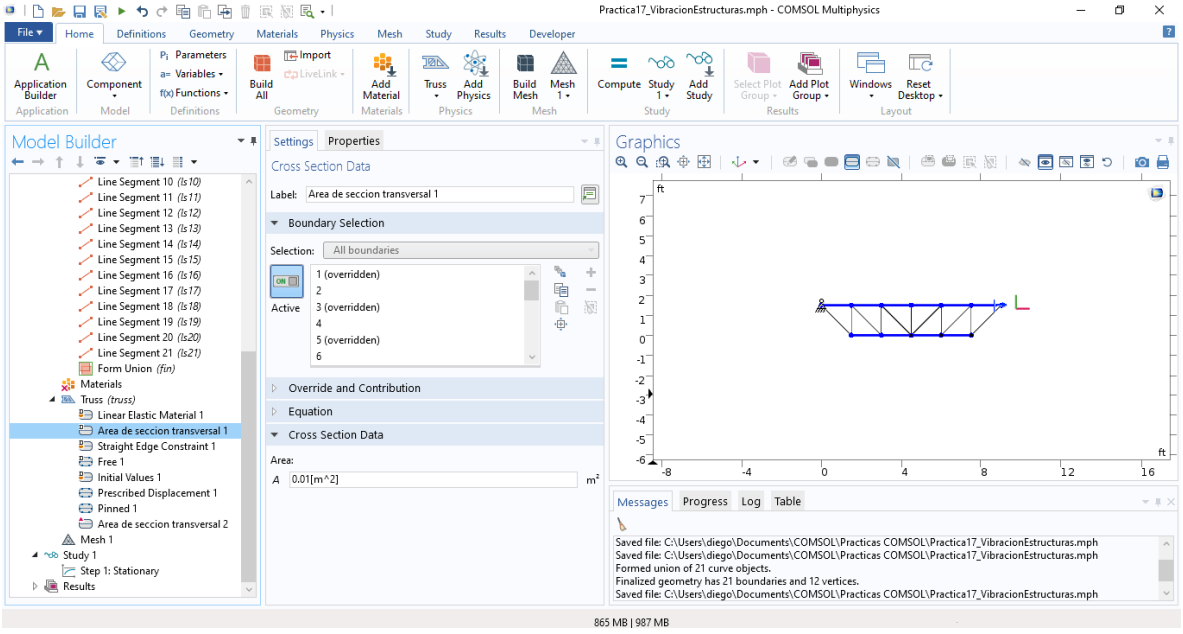

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Model Builder
Settings
Properties
Cross Section Data
Label: Cross Section Data 1
Boundary Selection
Selection: All boundaries
Active
Override and Contribution
Equation
Cross Section Data
Area:
A 0.01[m^2]
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
883 MB | 1009 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Cross Section Data
Label: Cross Section Data 1
Boundary Selection
Selection: All boundaries
1 (overridden)
2 (overridden)
3 (overridden)
4 (overridden)
5 (overridden)
6 (overridden)
Copy as Code to Clipboard
Copy
Duplicate Ctrl+Shift+D
Rename F2
Settings
Properties
Help F1
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
887 MB | 1005 MB

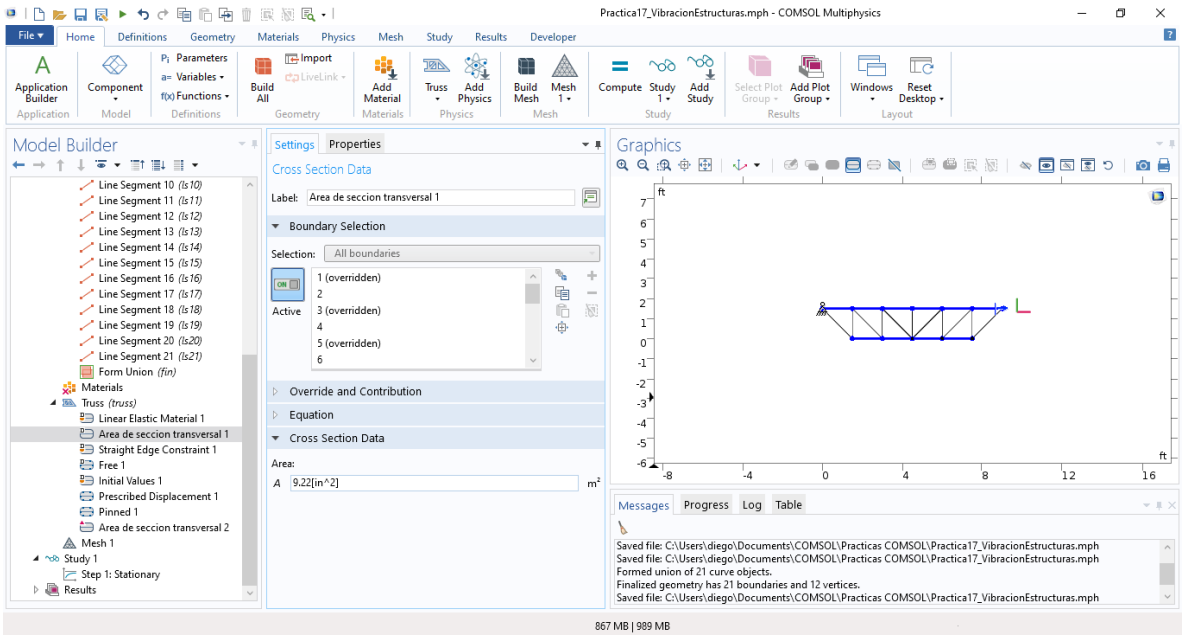

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Cross Section Data
Label: Area de seccion transversal 2
Boundary Selection
Selection: All boundaries
Active
Override and Contribution
Equation
Cross Section Data
Area:
A 0.01[m^2] m²
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
881 MB | 1003 MB

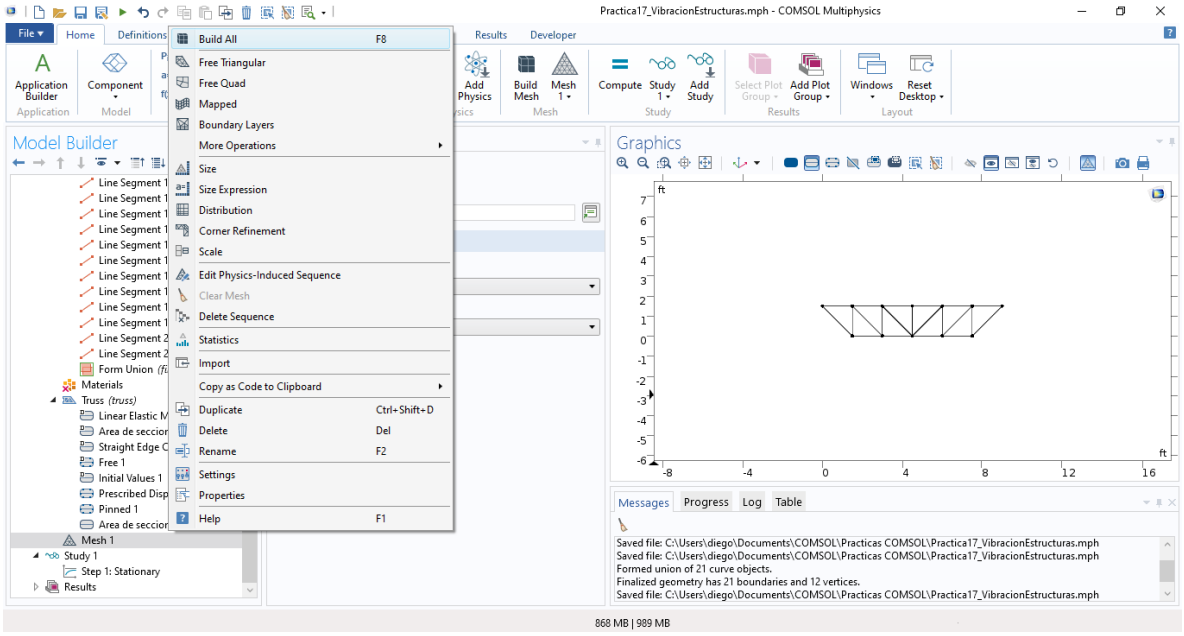

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Cross Section Data
Label: Area de seccion transversal 2
Boundary Selection
Selection: All boundaries
Active
Override and Contribution
Equation
Cross Section Data
Area:
A 4.61[in^2] m²
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
873 MB | 995 MB

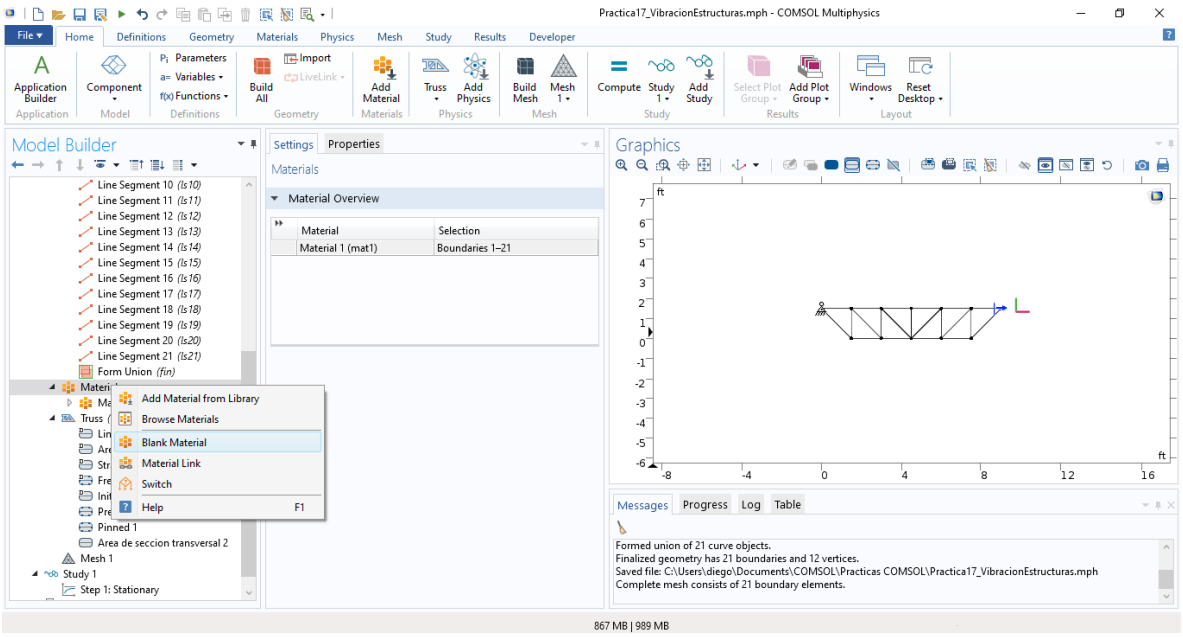


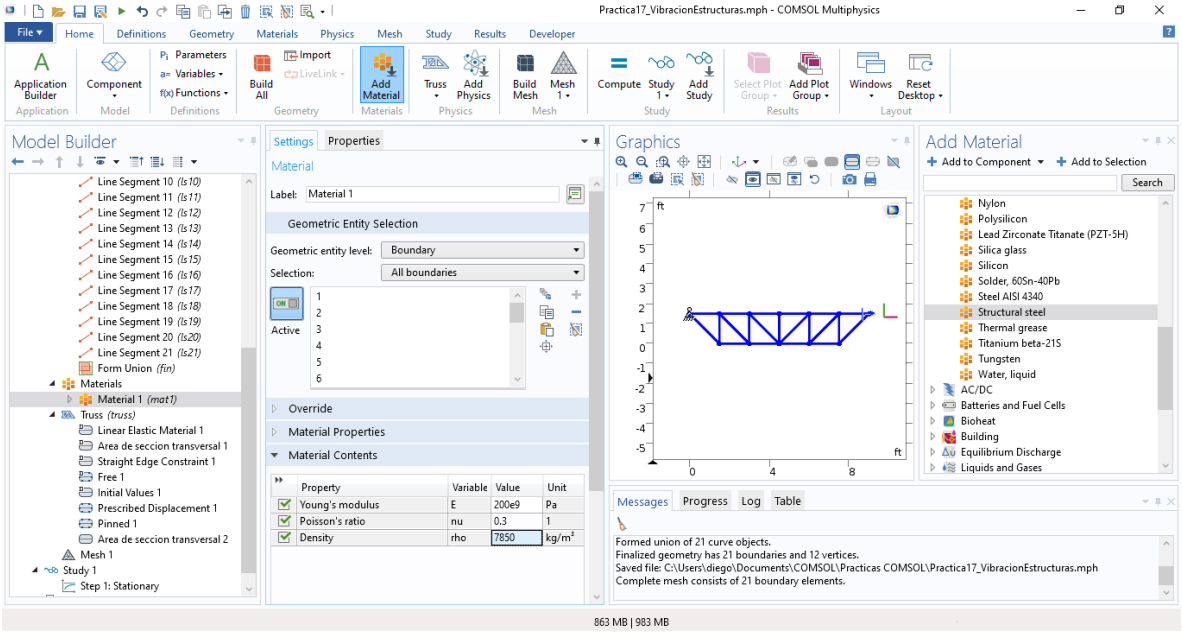


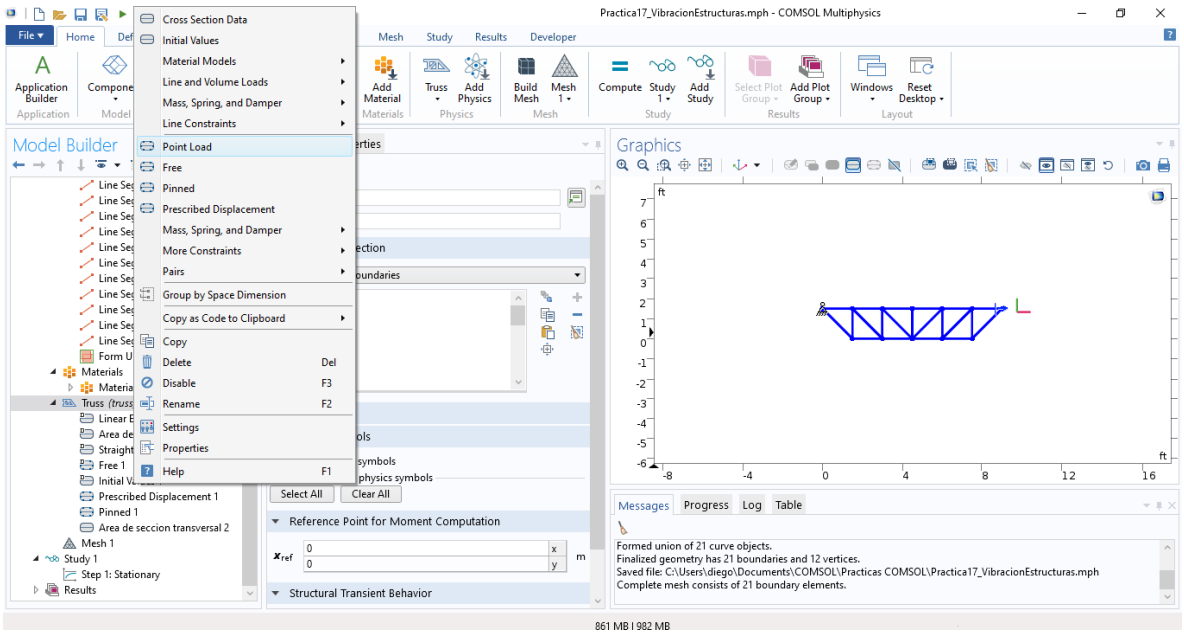
Va a haber 2 áreas de sección transversal distintas en la estructura, por eso se deben definir de la siguiente manera:

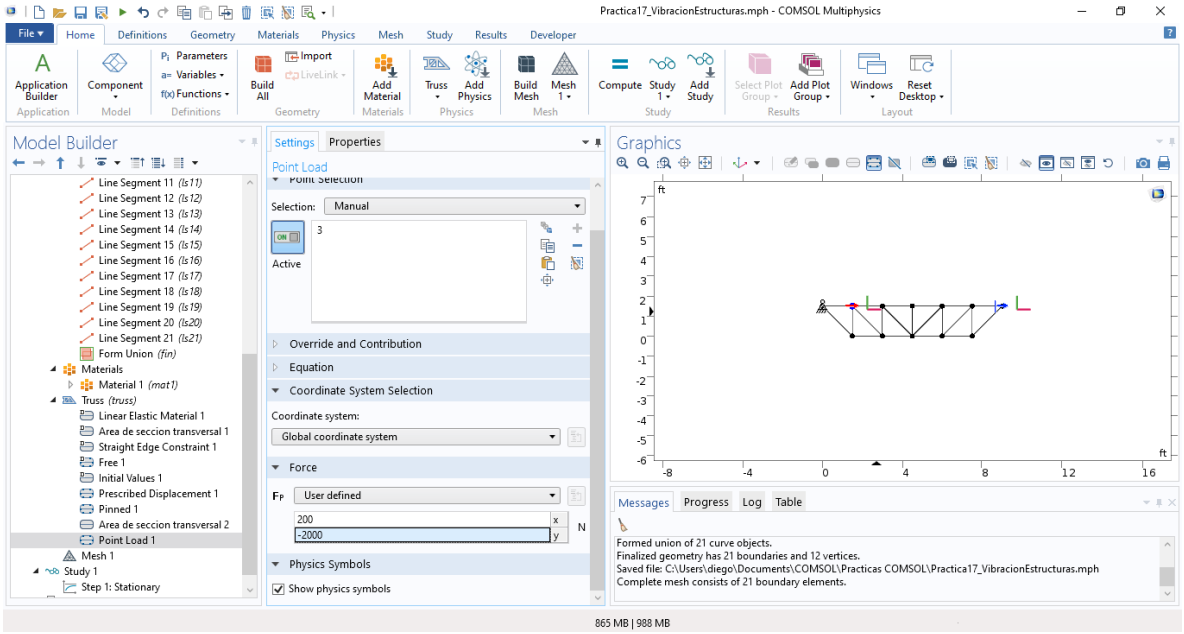

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Study 1
Add Study
Select Plot Group
Add Plot Group
Windows
Reset Desktop
Model Builder
Line Segment 10 (ls10)
Line Segment 11 (ls11)
Line Segment 12 (ls12)
Line Segment 13 (ls13)
Line Segment 14 (ls14)
Line Segment 15 (ls15)
Line Segment 16 (ls16)
Line Segment 17 (ls17)
Line Segment 18 (ls18)
Line Segment 19 (ls19)
Line Segment 20 (ls20)
Line Segment 21 (ls21)
Form Union (fin)
Materials
Truss (truss)
Linear Elastic Material 1
Area de seccion transversal 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Prescribed Displacement 1
Pinned 1
Area de seccion transversal 2
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Cross Section Data
Label: Area de seccion transversal 1
Boundary Selection
Selection: All boundaries
Active
1 (overridden)
2
3 (overridden)
4
5 (overridden)
6
Override and Contribution
Equation
Cross Section Data
Area:
A 9.22[in^2] m²
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
867 MB | 989 MB

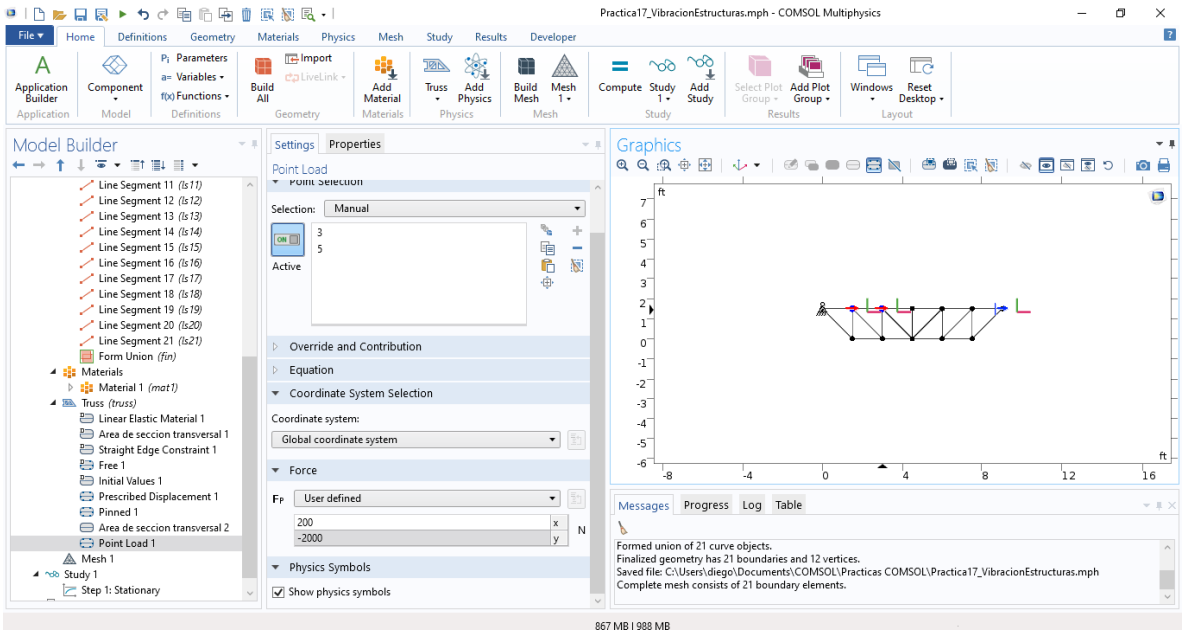

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Build All F8
Free Triangular
Free Quad
Mapped
Boundary Layers
More Operations
Size
Size Expression
Distribution
Corner Refinement
Scale
Edit Physics-Induced Sequence
Clear Mesh
Delete Sequence
Statistics
Import
Copy as Code to Clipboard
Duplicate Ctrl+Shift+D
Delete Del
Rename F2
Settings
Properties
Help F1
Model Builder
Graphics
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
868 MB | 989 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Study 1
Add Study
Select Plot Group
Add Plot Group
Windows
Reset Desktop
Model Builder
Line Segment 10 (ls10)
Line Segment 11 (ls11)
Line Segment 12 (ls12)
Line Segment 13 (ls13)
Line Segment 14 (ls14)
Line Segment 15 (ls15)
Line Segment 16 (ls16)
Line Segment 17 (ls17)
Line Segment 18 (ls18)
Line Segment 19 (ls19)
Line Segment 20 (ls20)
Line Segment 21 (ls21)
Form Union (fin)
Add Material from Library
Browse Materials
Blank Material
Material Link
Switch
Help F1
Pinned 1
Area de seccion transversal 2
Mesh 1
Study 1
Step 1: Stationary
Settings
Properties
Materials
Material Overview
Material
Selection
Material 1 (mat1)
Boundaries 1–21
Graphics
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
867 MB | 989 MB

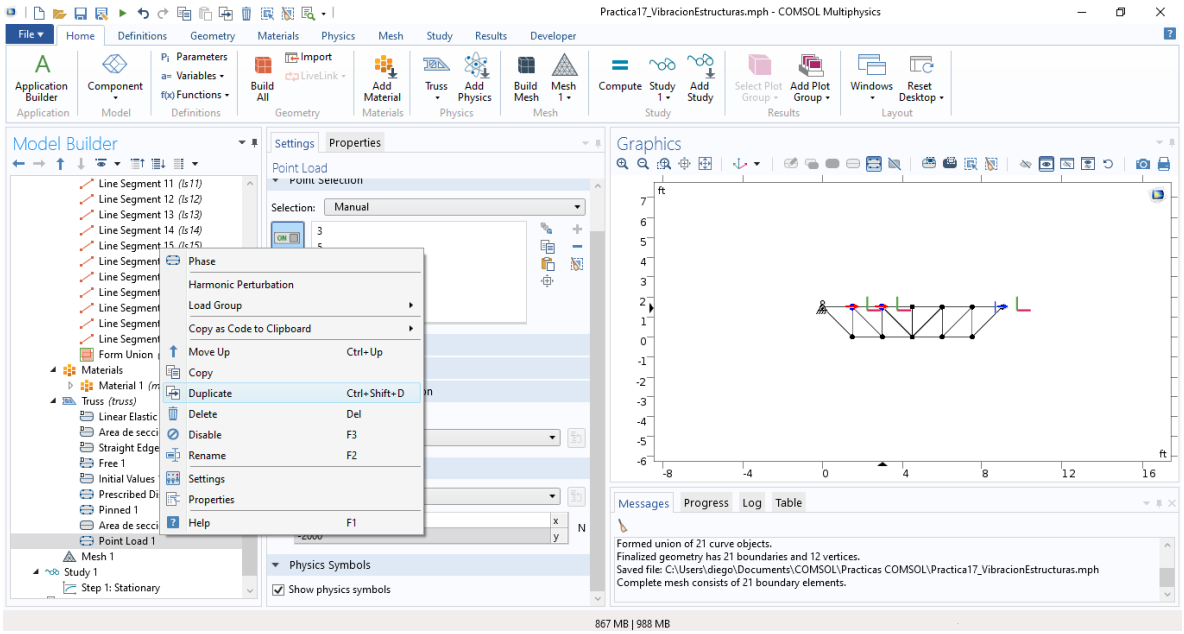

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Material 1 (mat1)
Truss (truss)
Linear Elastic Material 1
Area de seccion transversal 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Prescribed Displacement 1
Pinned 1
Area de seccion transversal 2
Mesh 1
Study 1
Step 1: Stationary
Settings
Properties
Material
Label: Material 1
Geometric Entity Selection
Geometric entity level: Boundary
Selection: All boundaries
Active
Override
Material Properties
Material Contents
Property
Variable
Value
Unit
Young's modulus
E
200e9
Pa
Poisson's ratio
nu
0.3
1
Density
rho
7850
kg/m³
Graphics
Add Material
Add to Component
Add to Selection
Search
Nylon
Polysilicon
Lead Zirconate Titanate (PZT-5H)
Silica glass
Silicon
Solder, 60Sn-40Pb
Steel AISI 4340
Structural steel
Thermal grease
Titanium beta-21S
Tungsten
Water, liquid
AC/DC
Batteries and Fuel Cells
Bioheat
Building
Equilibrium Discharge
Liquids and Gases
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
863 MB | 983 MB

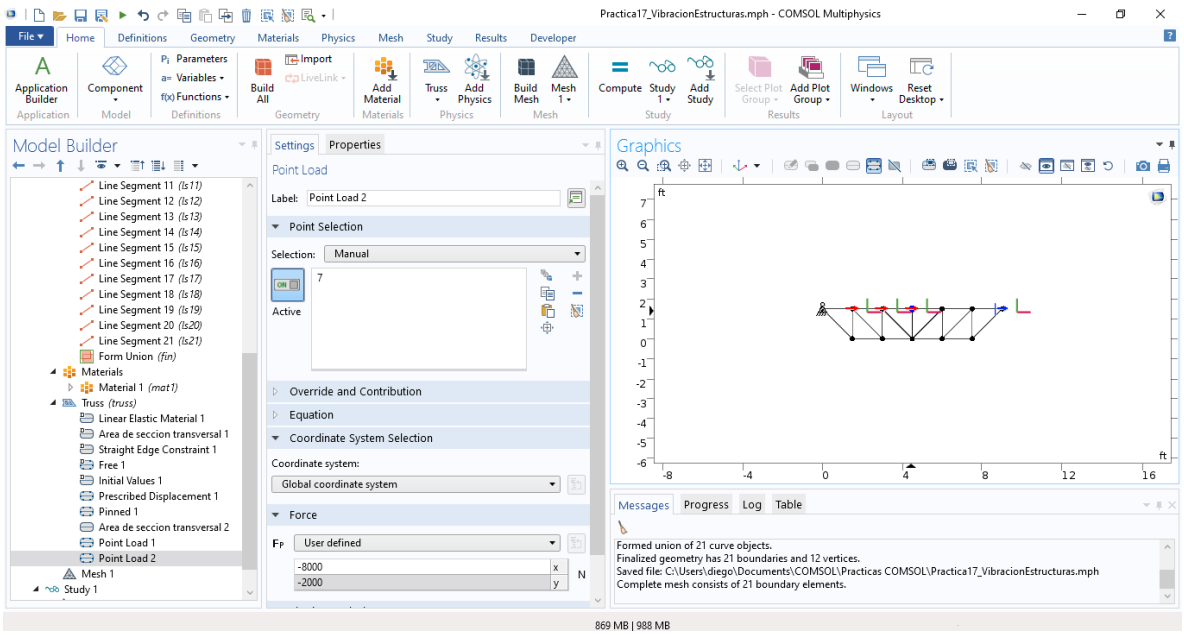

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Cross Section Data
Initial Values
Material Models
Line and Volume Loads
Mass, Spring, and Damper
Line Constraints
Point Load
Free
Pinned
Prescribed Displacement
Mass, Spring, and Damper
More Constraints
Pairs
Group by Space Dimension
Copy as Code to Clipboard
Copy
Delete Del
Disable F3
Rename F2
Settings
Properties
Help F1
Model Builder
Graphics
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
Reference Point for Moment Computation
Structural Transient Behavior
861 MB | 982 MB

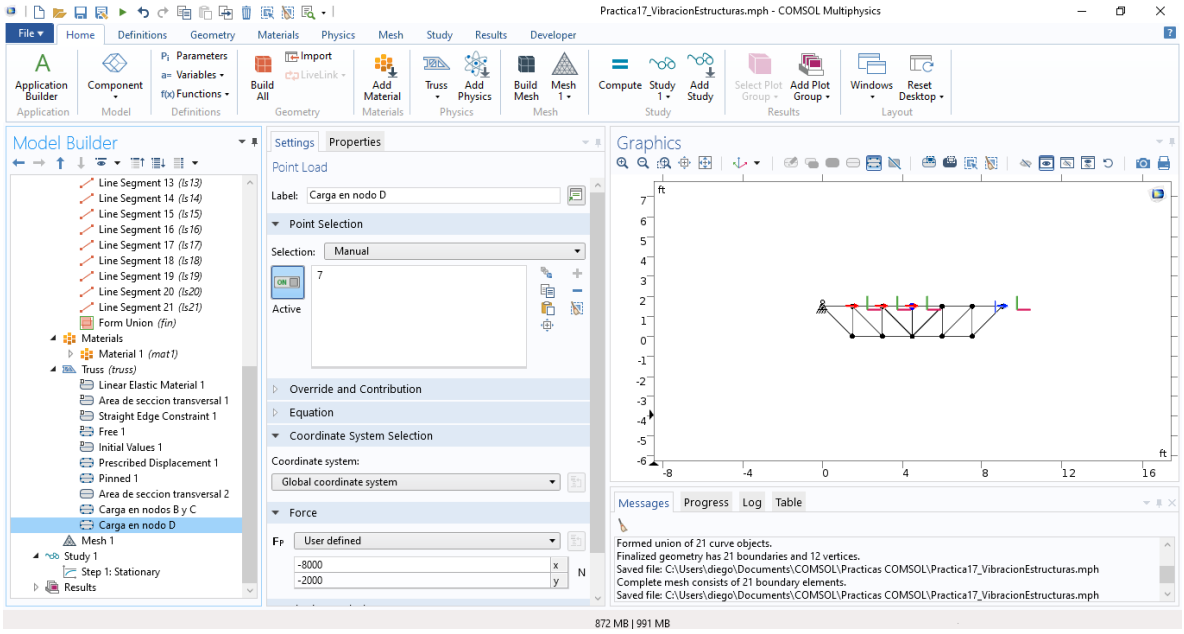

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Point Load
Selection: Manual
Active
Override and Contribution
Equation
Coordinate System Selection
Coordinate system:
Global coordinate system
Force
User defined
200
-2000
N
Physics Symbols
Show physics symbols
Graphics
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
865 MB | 988 MB

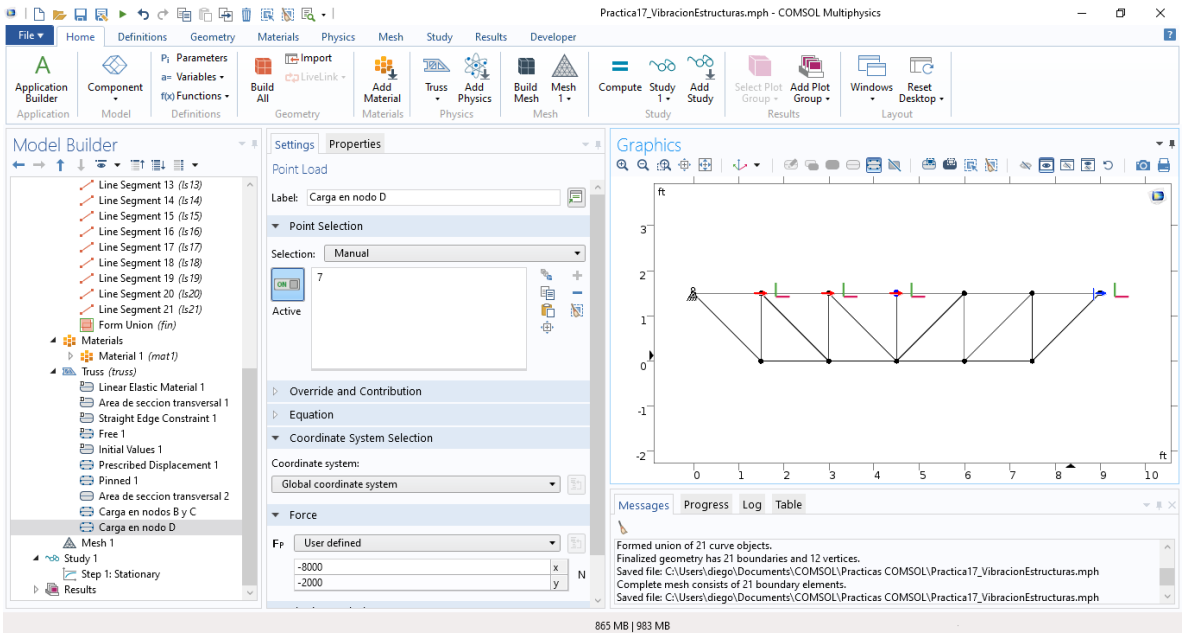


Como las cargas en estos nodos son iguales, las puedo aplicar el mismo Load Point de ambos puntos.

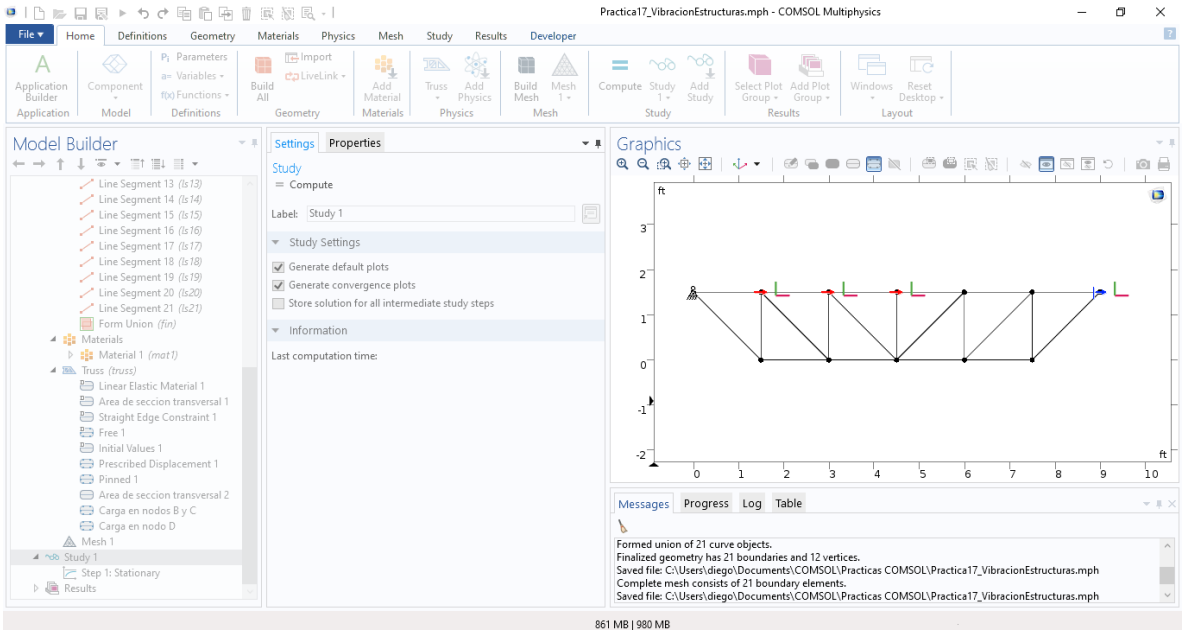


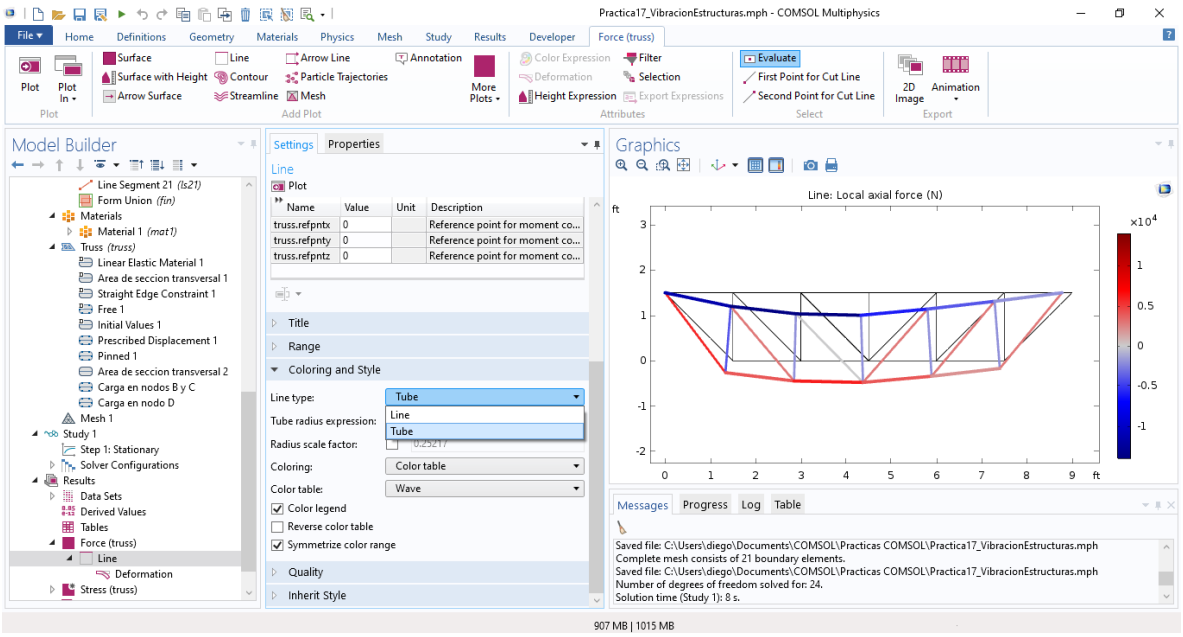

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Study 1
Add Study
Select Plot Group
Add Plot Group
Windows
Reset Desktop
Model Builder
Line Segment 11 (ls11)
Line Segment 12 (ls12)
Line Segment 13 (ls13)
Line Segment 14 (ls14)
Line Segment 15 (ls15)
Line Segment 16 (ls16)
Line Segment 17 (ls17)
Line Segment 18 (ls18)
Line Segment 19 (ls19)
Line Segment 20 (ls20)
Line Segment 21 (ls21)
Form Union (fin)
Materials
Material 1 (mat1)
Truss (truss)
Linear Elastic Material 1
Area de seccion transversal 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Prescribed Displacement 1
Pinned 1
Area de seccion transversal 2
Point Load 1
Point Load 2
Mesh 1
Study 1
Settings
Properties
Point Load
Label: Point Load 2
Point Selection
Selection: Manual
Active
7
Override and Contribution
Equation
Coordinate System Selection
Coordinate system:
Global coordinate system
Force
Fp
User defined
-8000
-2000
N
Graphics
ft
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
869 MB | 988 MB

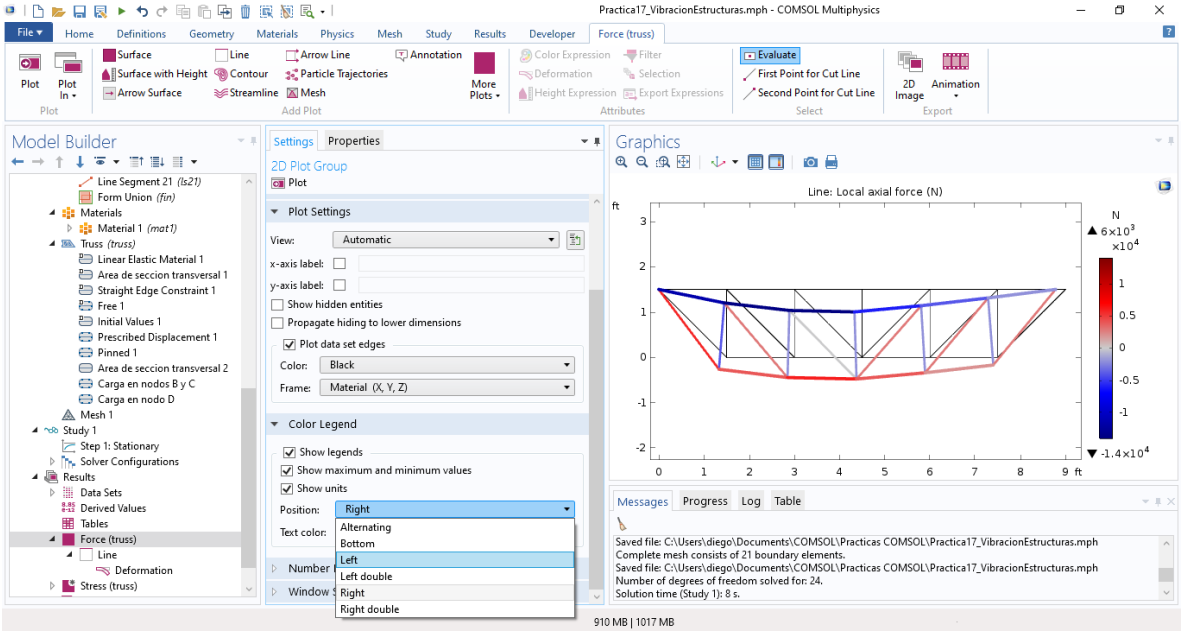

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Study 1
Add Study
Select Plot Group
Add Plot Group
Windows
Reset Desktop
Model Builder
Line Segment 13 (ls13)
Line Segment 14 (ls14)
Line Segment 15 (ls15)
Line Segment 16 (ls16)
Line Segment 17 (ls17)
Line Segment 18 (ls18)
Line Segment 19 (ls19)
Line Segment 20 (ls20)
Line Segment 21 (ls21)
Form Union (fin)
Materials
Material 1 (mat1)
Truss (truss)
Linear Elastic Material 1
Area de seccion transversal 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Prescribed Displacement 1
Pinned 1
Area de seccion transversal 2
Carga en nodos B y C
Carga en nodo D
Mesh 1
Study 1
Step 1: Stationary
Results
Settings
Properties
Point Load
Label: Carga en nodo D
Point Selection
Selection: Manual
Active
7
Override and Contribution
Equation
Coordinate System Selection
Coordinate system:
Global coordinate system
Force
Fp
User defined
-8000
-2000
N
Graphics
ft
Messages
Progress
Log
Table
Formed union of 21 curve objects.
Finalized geometry has 21 boundaries and 12 vertices.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
872 MB | 991 MB

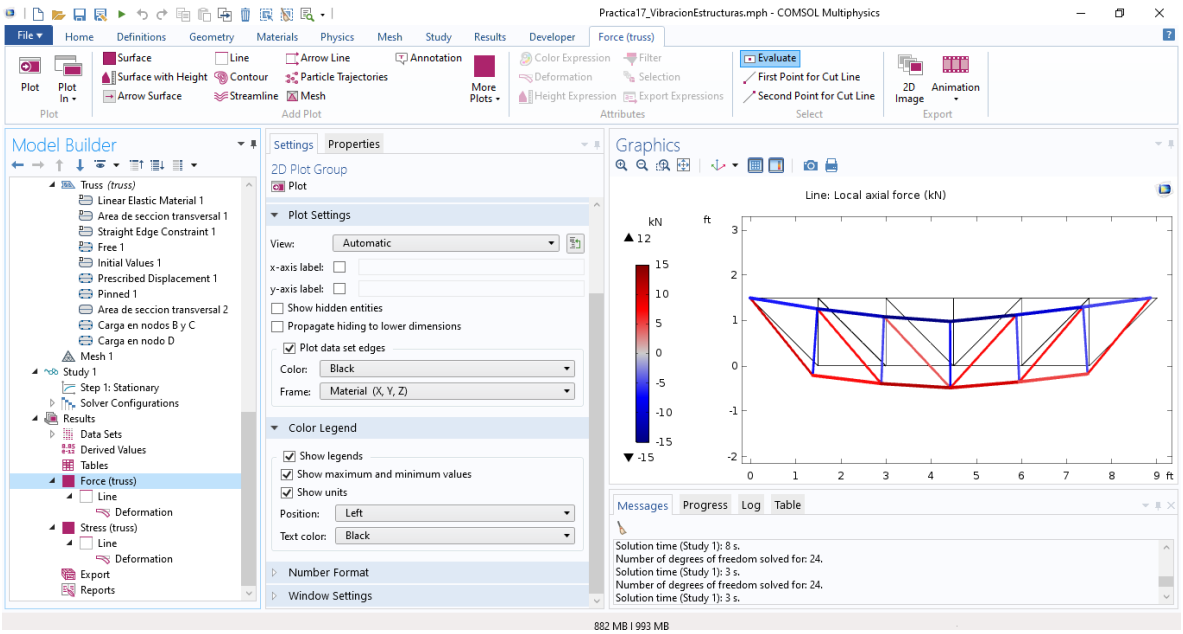


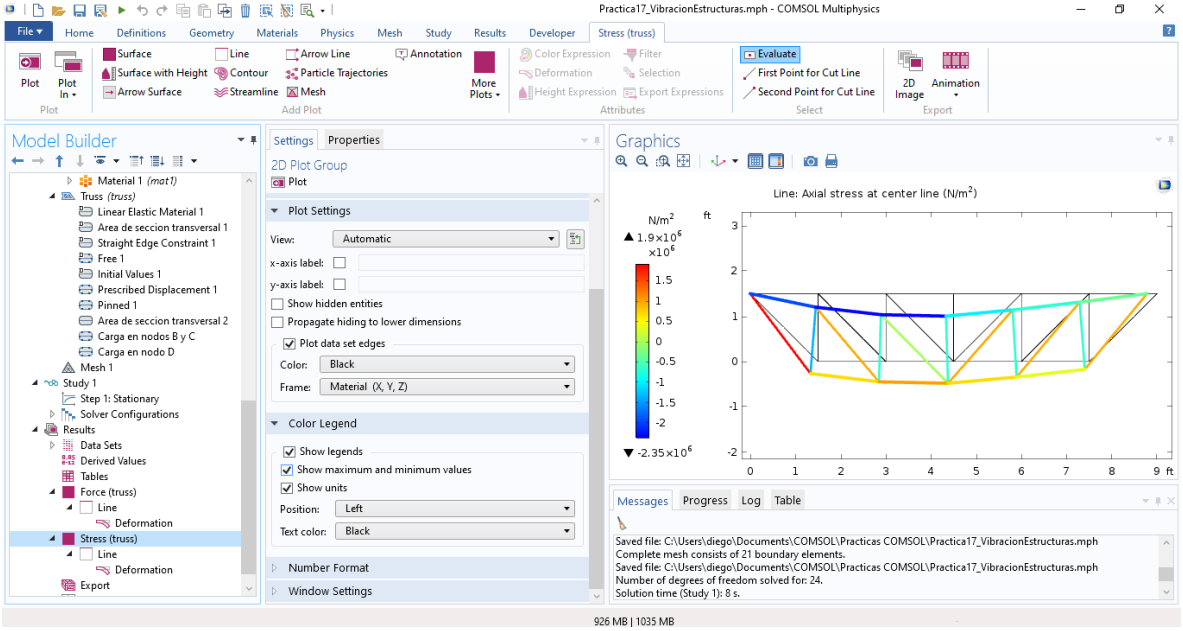
RESULTADO DEL ELEMENTO FINITO EN COMSOL:

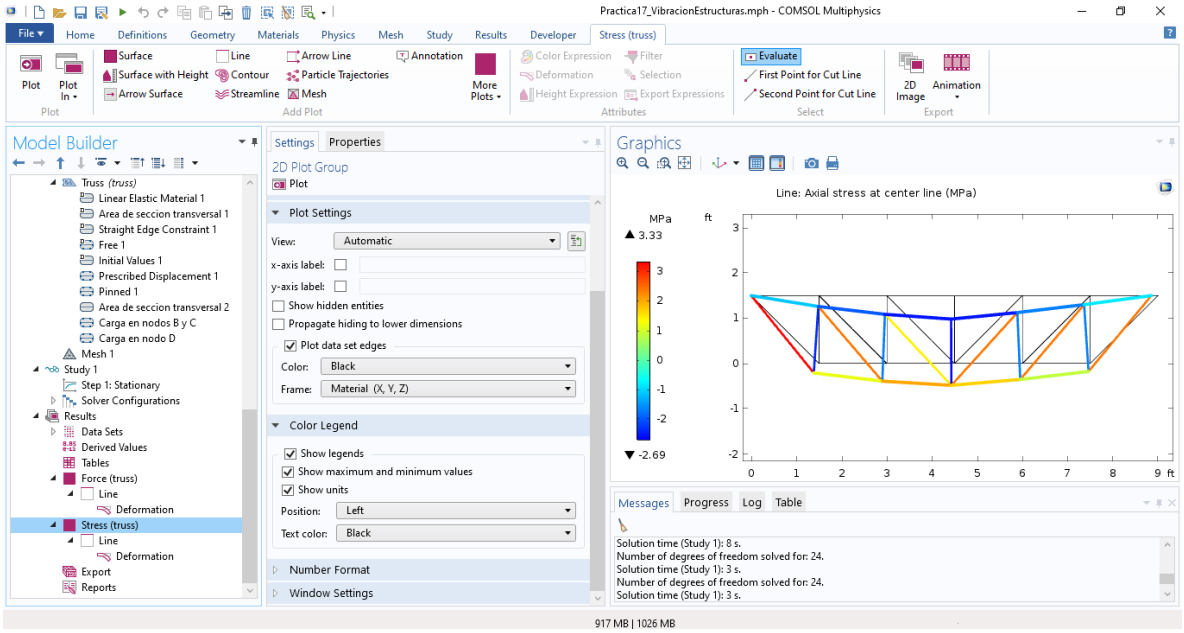



Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Force (truss)
Plot
Plot In
Surface
Surface with Height
Arrow Surface
Line
Contour
Streamline
Arrow Line
Particle Trajectories
Mesh
Annotation
More Plots
Color Expression
Deformation
Height Expression
Filter
Selection
Export Expressions
Evaluate
First Point for Cut Line
Second Point for Cut Line
2D Image
Animation
Model Builder
Line Segment 21 (ls21)
Form Union (fin)
Materials
Material 1 (mat1)
Truss (truss)
Linear Elastic Material 1
Area de seccion transversal 1
Straight Edge Constraint 1
Free 1
Initial Values 1
Prescribed Displacement 1
Pinned 1
Area de seccion transversal 2
Carga en nodos B y C
Carga en nodo D
Mesh 1
Study 1
Step 1: Stationary
Solver Configurations
Results
Data Sets
Derived Values
Tables
Force (truss)
Line
Deformation
Stress (truss)
Settings
Properties
Line
Plot
Name
Value
Unit
Description
truss.refpntx
truss.refpnty
truss.refpntz
Reference point for moment co...
Title
Range
Coloring and Style
Line type:
Tube
Line
Tube radius expression:
Radius scale factor:
Coloring:
Color table
Color table:
Wave
Color legend
Reverse color table
Symmetrize color range
Quality
Inherit Style
Graphics
Line: Local axial force (N)
ft
×10^4
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 8 s.
907 MB | 1015 MB

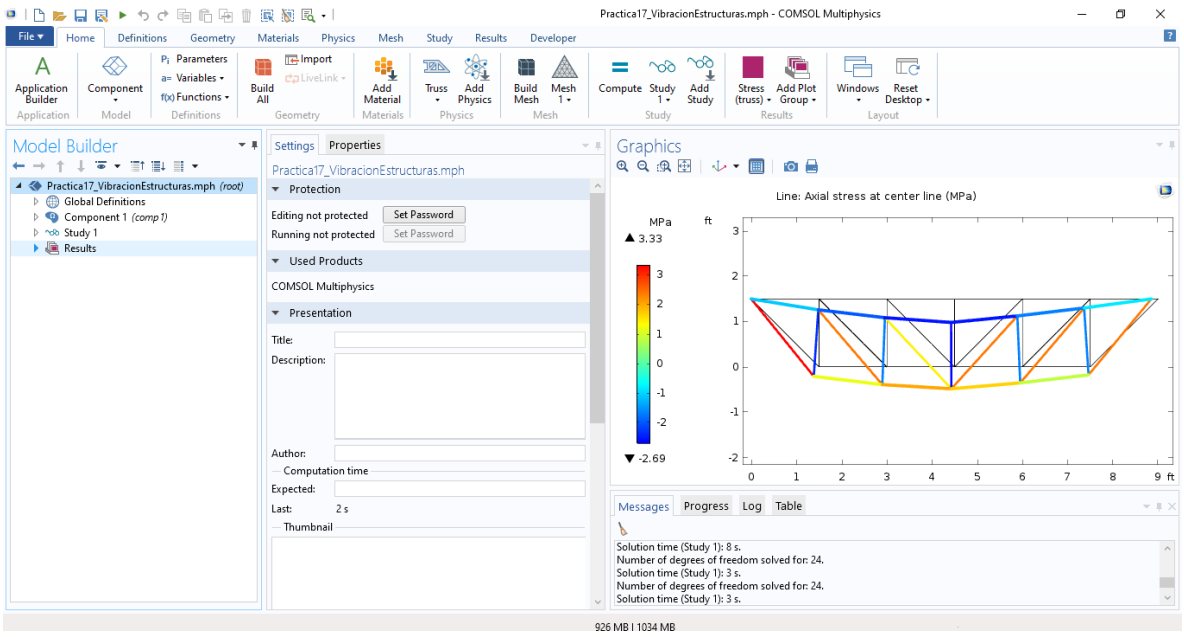

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Plot
Plot Settings
View:
Automatic
x-axis label:
y-axis label:
Show hidden entities
Propagate hiding to lower dimensions
Plot data set edges
Color:
Black
Frame:
Material (X, Y, Z)
Color Legend
Show legends
Show maximum and minimum values
Show units
Position:
Right
Text color:
Alternating
Bottom
Left
Left double
Right
Right double
Graphics
Line: Local axial force (N)
N
▲ 6×10^3
×10^4
▼ -1.4×10^4
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 8 s.
910 MB | 1017 MB

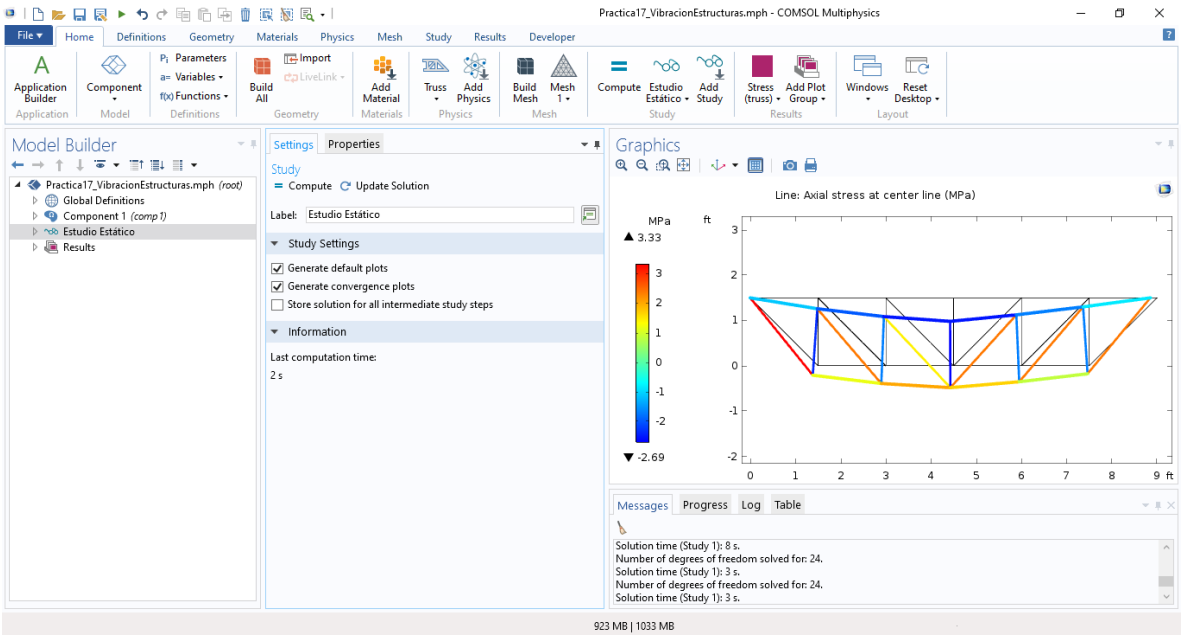

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Force (truss)
Model Builder
Graphics
Line: Local axial force (kN)
Messages
Progress
Log
Table
Solution time (Study 1): 8 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
882 MB | 993 MB

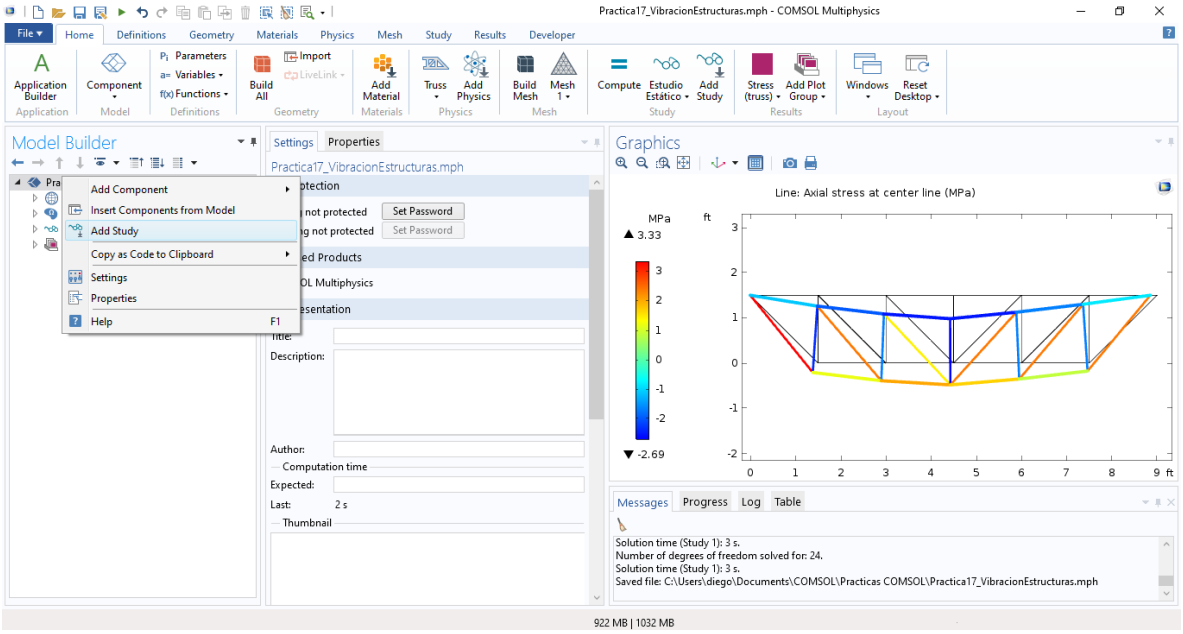

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Stress (truss)
Model Builder
Graphics
Line: Axial stress at center line (N/m²)
Messages
Progress
Log
Table
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Complete mesh consists of 21 boundary elements.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 8 s.
926 MB | 1035 MB

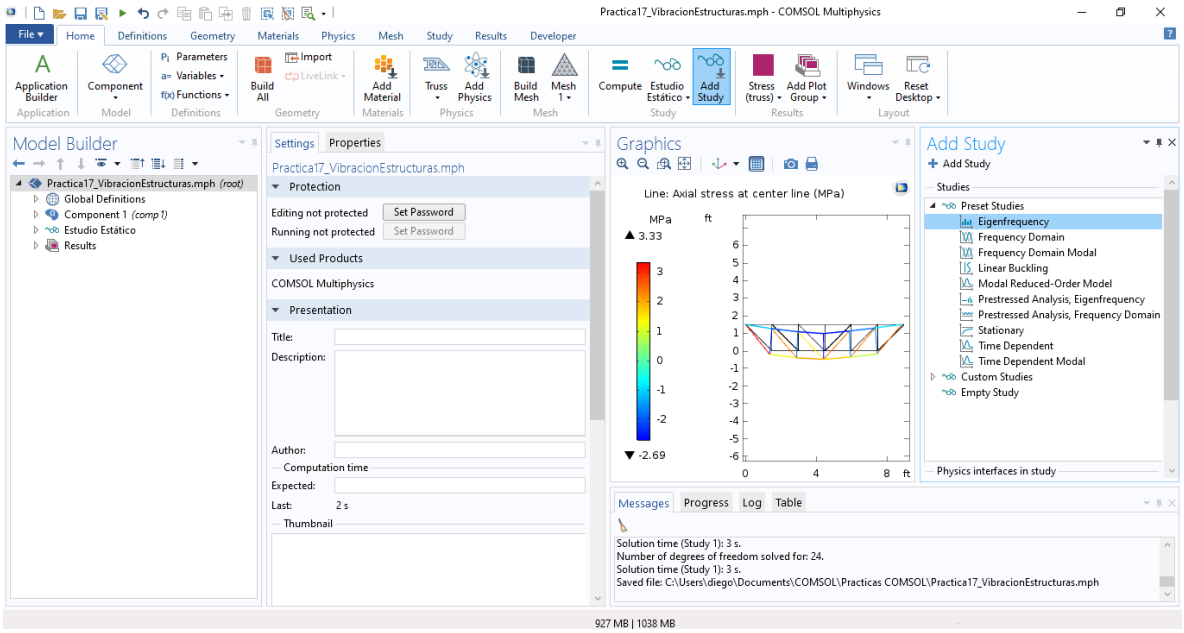


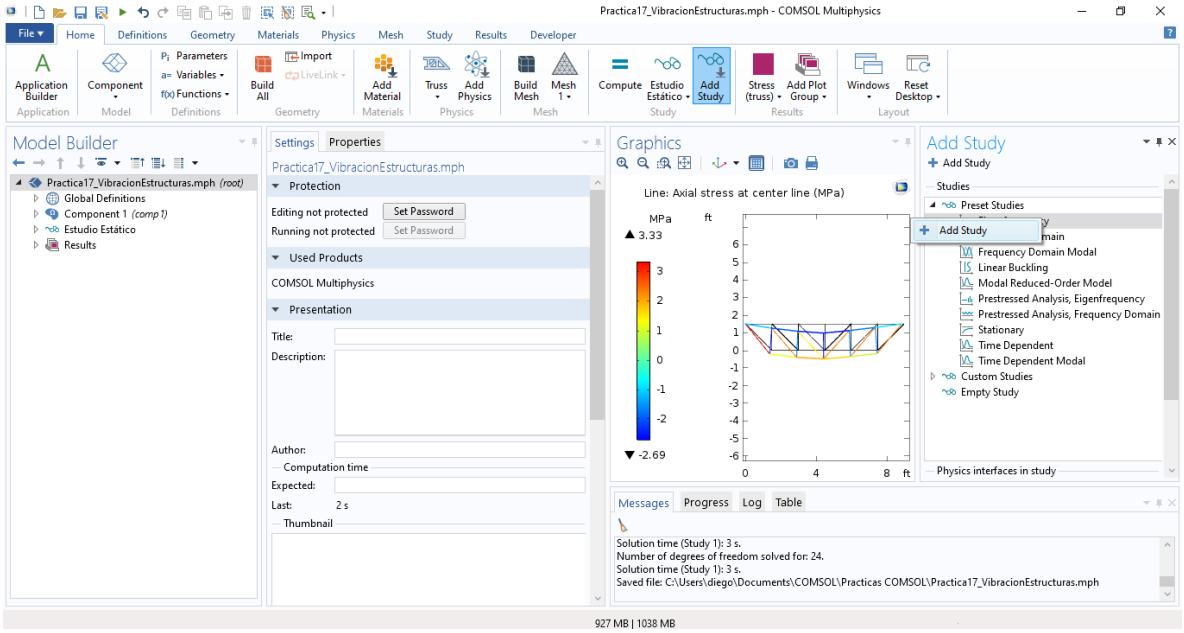
Todas las pestañas de la izquierda las minimizo para que agregue otro estudio, que será el de las vibraciones armónicas.

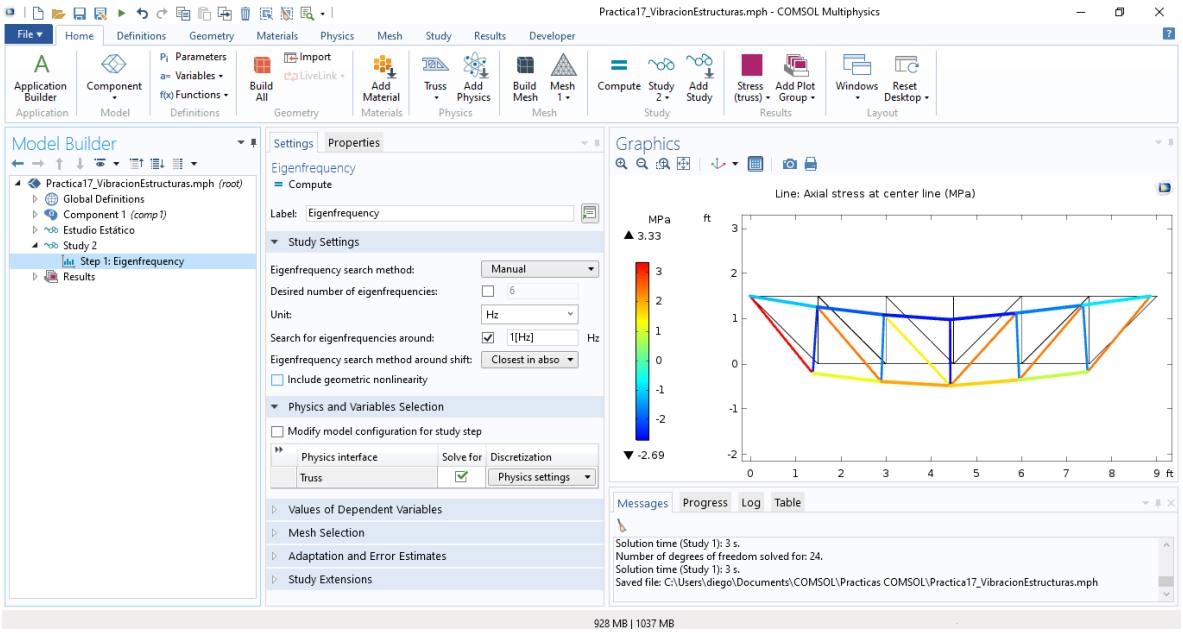



Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Estudio Estático
Add Study
Stress (truss)
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Results
Settings
Properties
Study
Compute
Update Solution
Label: Estudio Estático
Study Settings
Generate default plots
Generate convergence plots
Store solution for all intermediate study steps
Information
Last computation time:
2 s
Graphics
Line: Axial stress at center line (MPa)
MPa
ft
▲ 3.33
▼ -2.69
Messages
Progress
Log
Table
Solution time (Study 1): 8 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
923 MB | 1033 MB

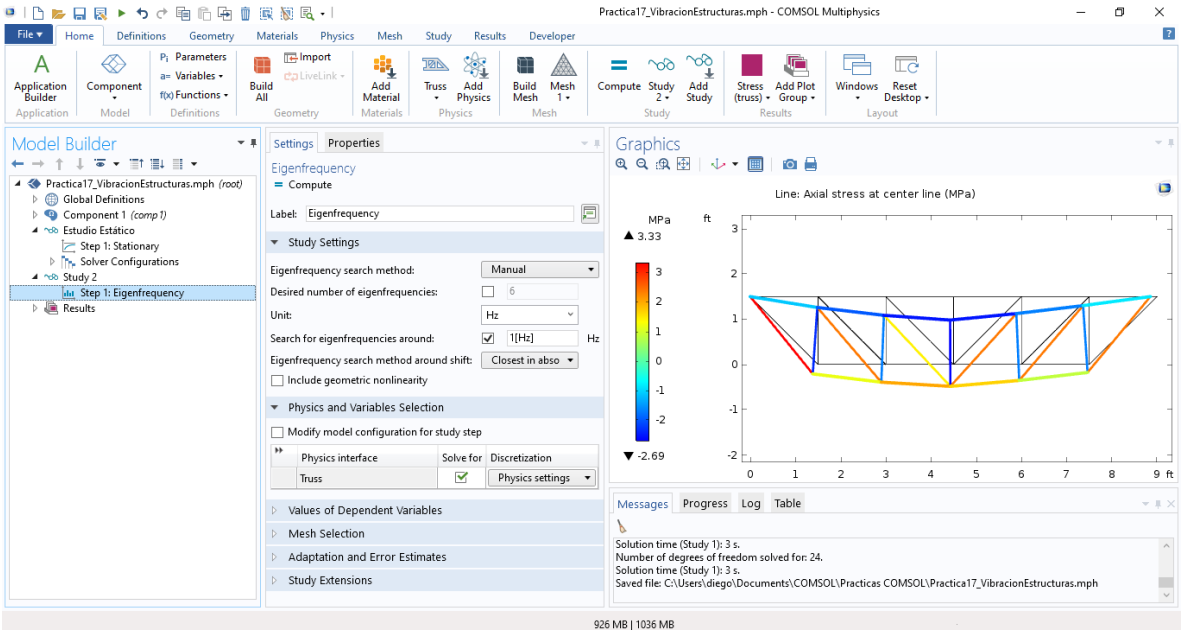

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Add Component
Insert Components from Model
Add Study
Copy as Code to Clipboard
Settings
Properties
Help
F1
Practica17_VibracionEstructuras.mph
Set Password
Description:
Author:
Computation time
Expected:
Last: 2 s
Thumbnail
Graphics
Line: Axial stress at center line (MPa)
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
922 MB | 1032 MB

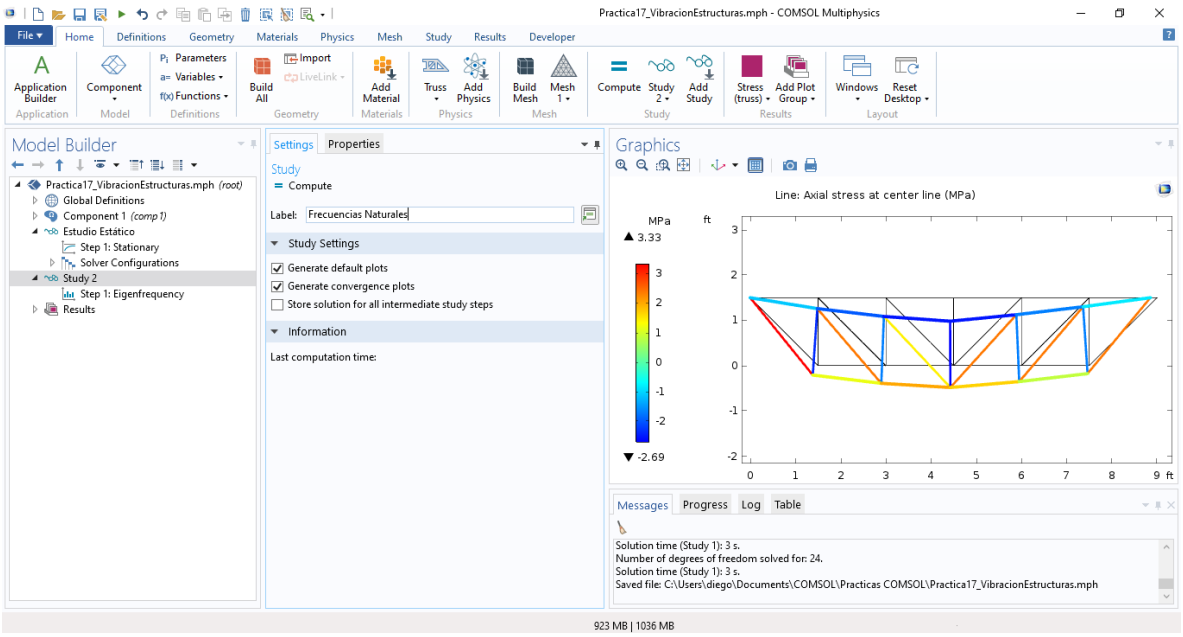

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Estudio Estático
Add Study
Stress (truss)
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Results
Settings
Properties
Practica17_VibracionEstructuras.mph
Protection
Editing not protected
Set Password
Running not protected
Set Password
Used Products
COMSOL Multiphysics
Presentation
Title:
Description:
Author:
Computation time
Expected:
Last: 2 s
Thumbnail
Graphics
Line: Axial stress at center line (MPa)
MPa
ft
▲ 3.33
▼ -2.69
Add Study
Add Study
Studies
Preset Studies
Eigenfrequency
Frequency Domain
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
927 MB | 1038 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Estudio Estático
Add Study
Stress (truss)
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Results
Settings
Properties
Practica17_VibracionEstructuras.mph
Protection
Editing not protected
Set Password
Running not protected
Set Password
Used Products
COMSOL Multiphysics
Presentation
Title:
Description:
Author:
Computation time
Expected:
Last: 2 s
Thumbnail
Graphics
Line: Axial stress at center line (MPa)
MPa
ft
▲ 3.33
▼ -2.69
Add Study
Add Study
Studies
Preset Studies
Add Study
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
927 MB | 1038 MB

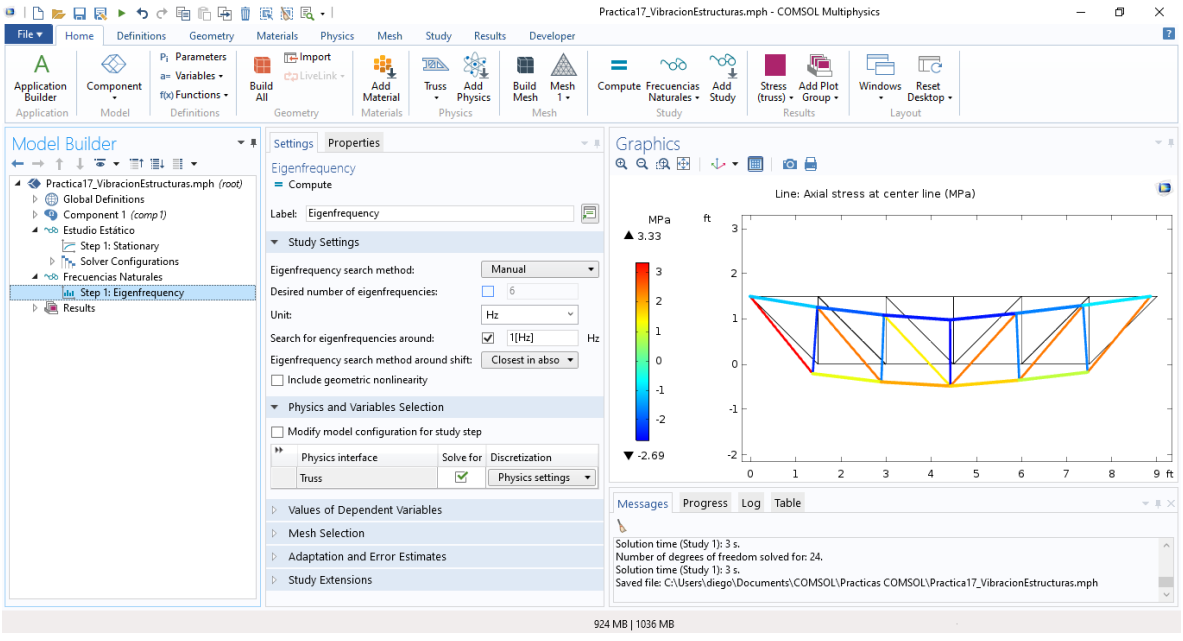


Ya con esto tengo 2 estudios, uno estacionario y otro que será el que medirá la frecuencia natural de la estructura y será transitoria.

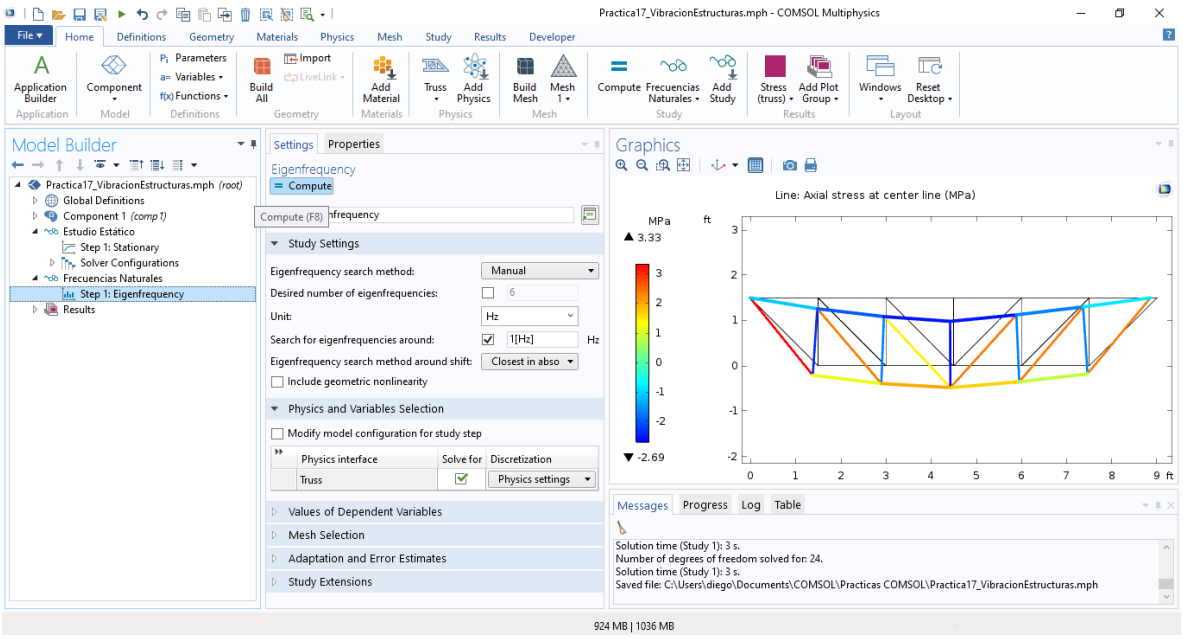




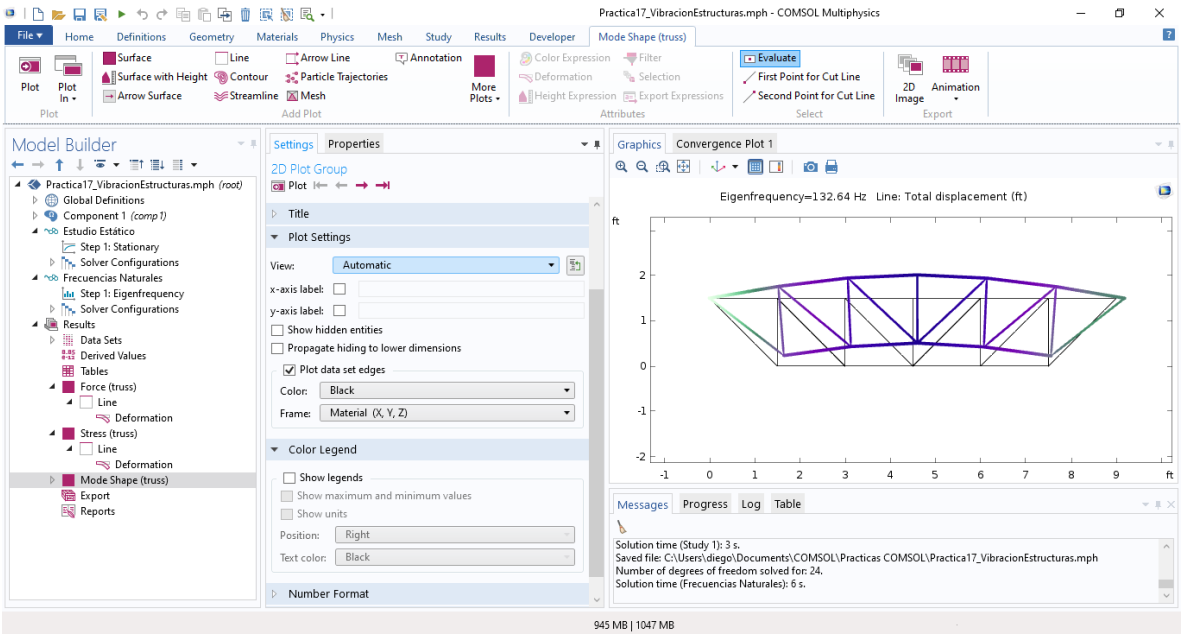
A este estudio de las propiedades de las vibraciones del sistema no le importa las cargas que se estén aplicando, las frecuencias naturales es una propiedad del sistema, así como su masa, su volumen, etc.

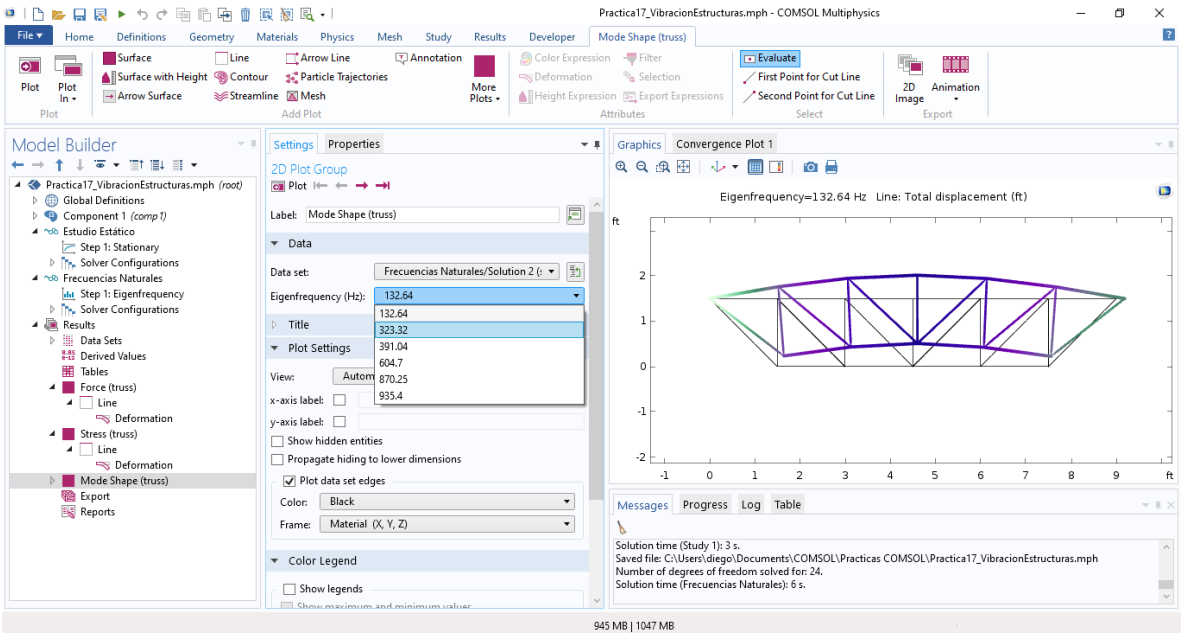


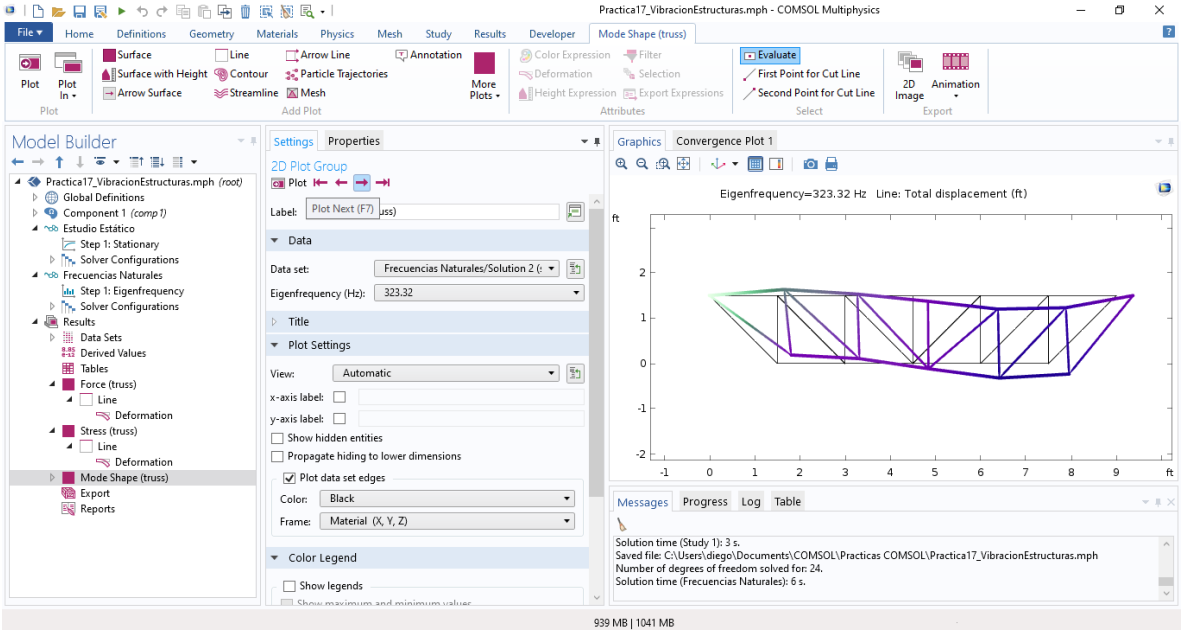
Por default el programa calcula las primeras 6 frecuencias naturales, ya que las más importantes son las primeras.

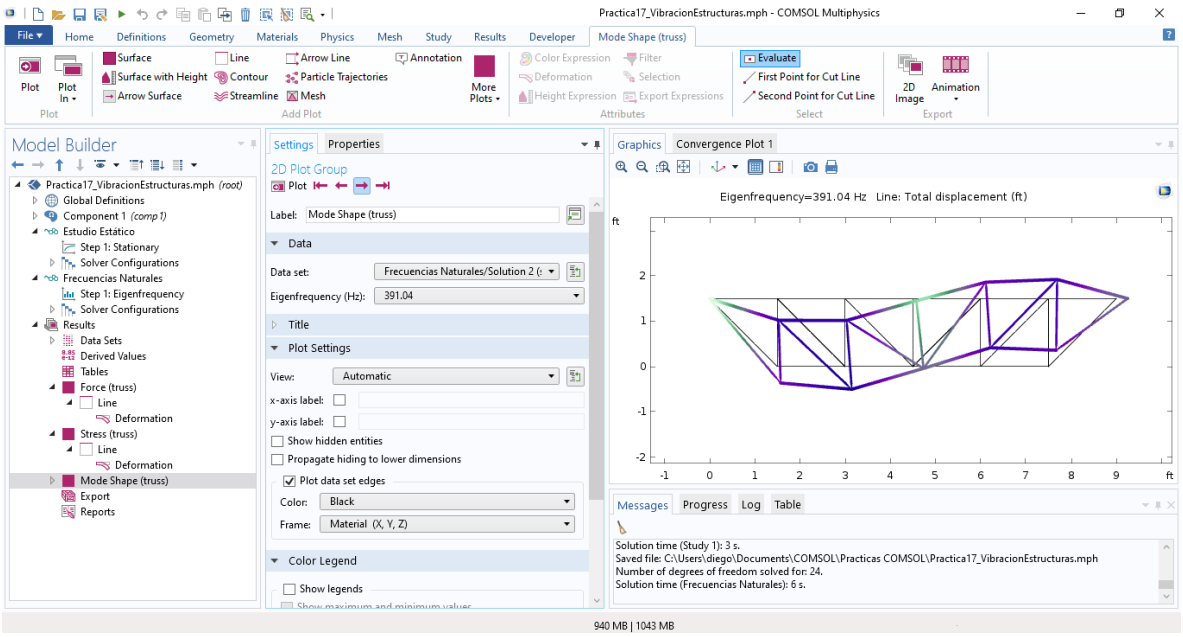


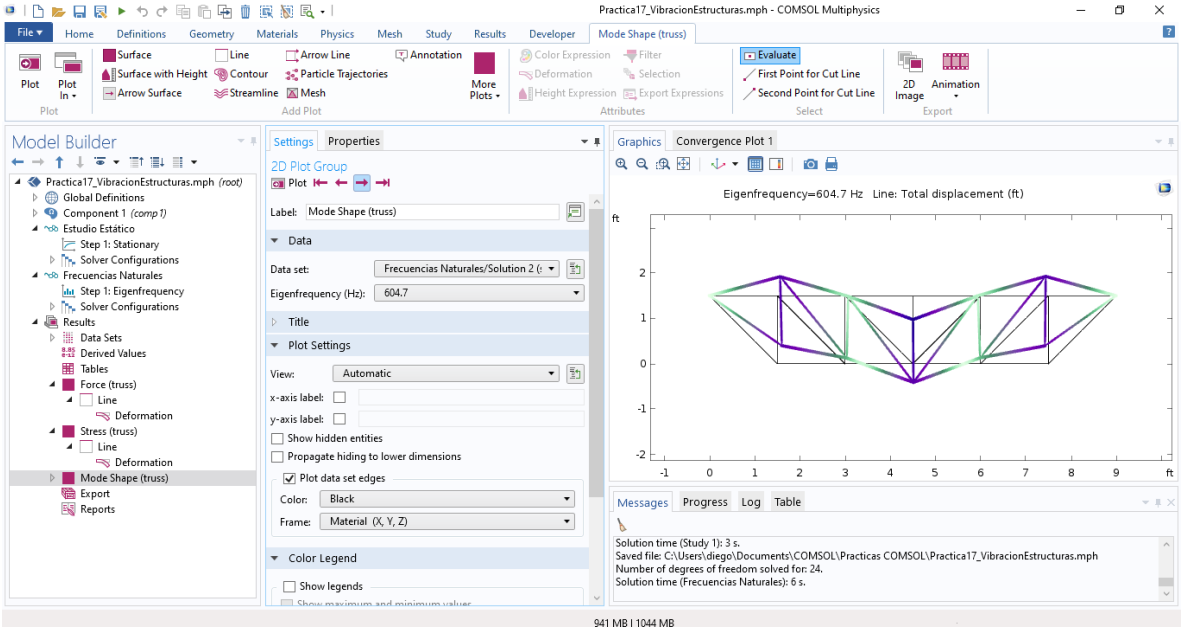
ANÁLISIS DE VIBRACIONES ARMÓNICAS EN COMSOL:

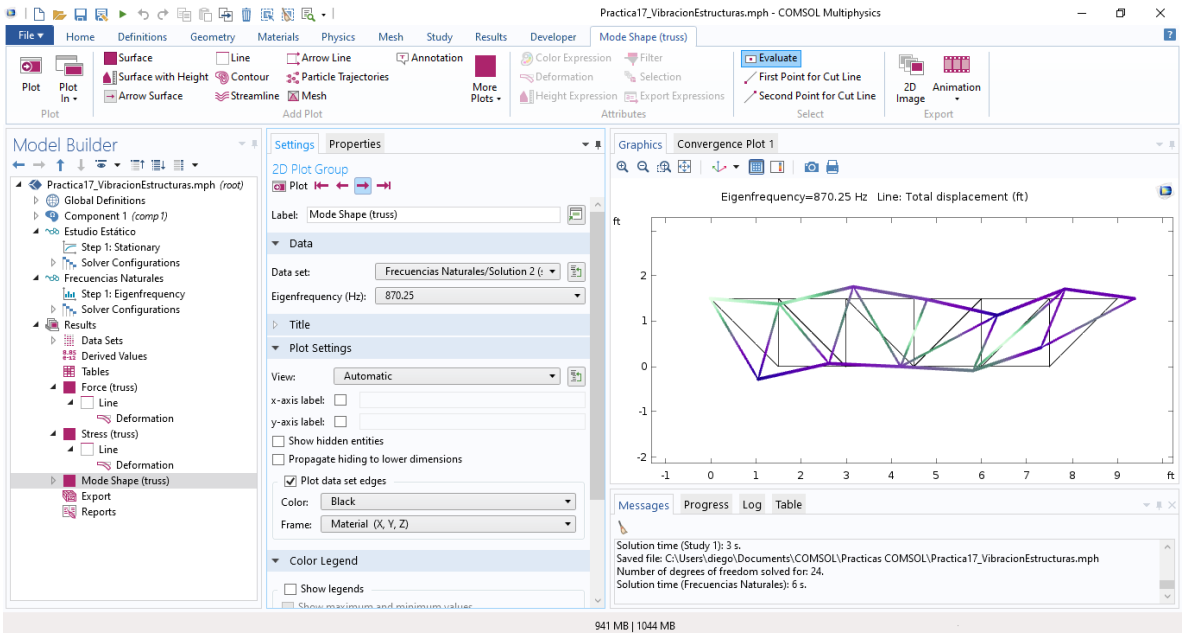


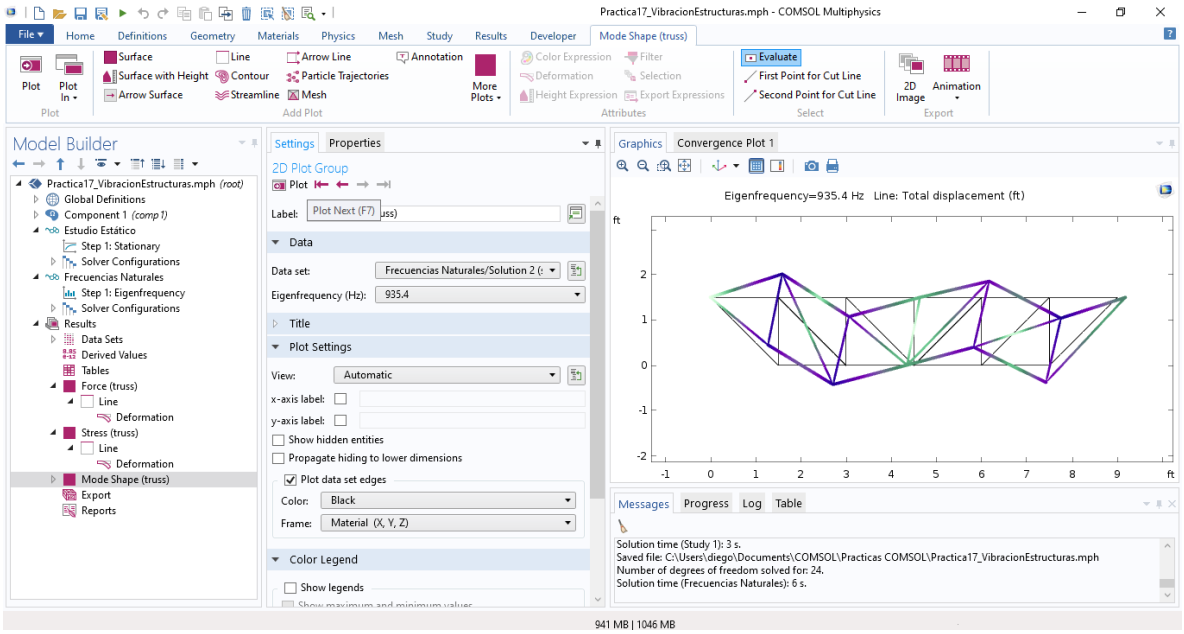

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Label: Mode Shape (truss)
Data
Data set: Frecuencias Naturales/Solution 2
Eigenfrequency (Hz): 132.64
132.64
323.32
391.04
604.7
870.25
935.4
Title
Plot Settings
Color Legend
Graphics
Convergence Plot 1
Eigenfrequency=132.64 Hz Line: Total displacement (ft)
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
945 MB | 1047 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Label: Plot Next (F7)
Data
Data set: Frecuencias Naturales/Solution 2
Eigenfrequency (Hz): 323.32
Title
Plot Settings
Color Legend
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz Line: Total displacement (ft)
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
939 MB | 1041 MB

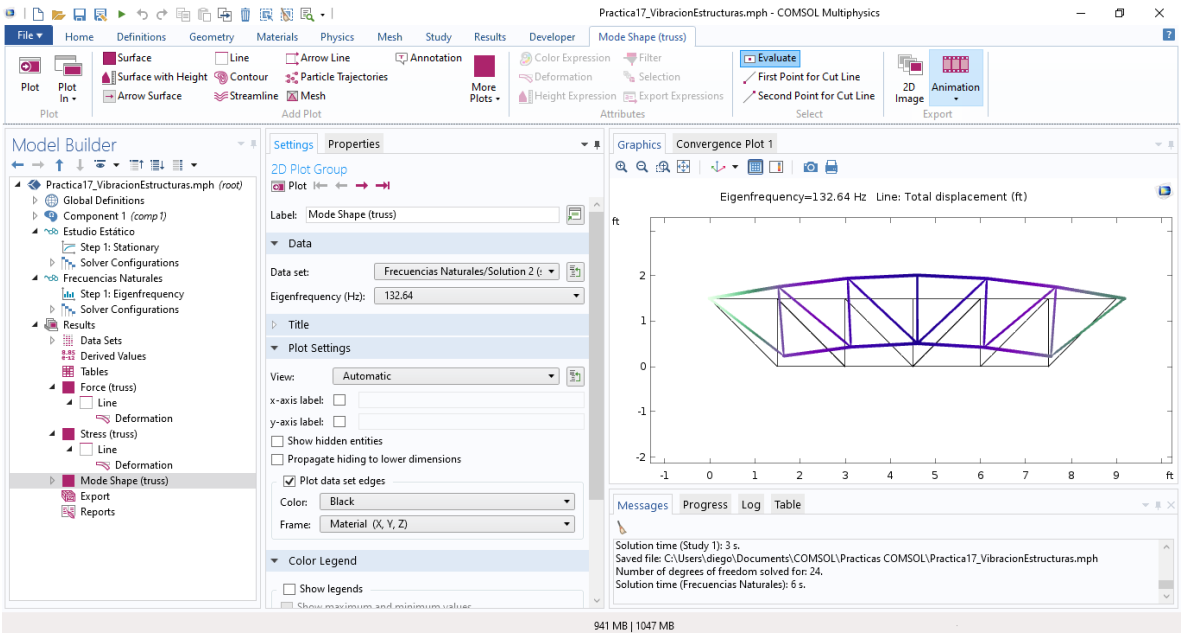

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Step 1: Stationary
Solver Configurations
Frecuencias Naturales
Step 1: Eigenfrequency
Solver Configurations
Results
Data Sets
Derived Values
Tables
Force (truss)
Line
Deformation
Stress (truss)
Line
Deformation
Mode Shape (truss)
Export
Reports
Settings
Properties
2D Plot Group
Label: Mode Shape (truss)
Data
Data set: Frecuencias Naturales/Solution 2
Eigenfrequency (Hz): 391.04
Title
Plot Settings
View: Automatic
x-axis label:
y-axis label:
Show hidden entities
Propagate hiding to lower dimensions
Plot data set edges
Color: Black
Frame: Material (X, Y, Z)
Color Legend
Show legends
Graphics
Convergence Plot 1
Eigenfrequency=391.04 Hz Line: Total displacement (ft)
ft
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
940 MB | 1043 MB


Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Step 1: Stationary
Solver Configurations
Frecuencias Naturales
Step 1: Eigenfrequency
Solver Configurations
Results
Data Sets
Derived Values
Tables
Force (truss)
Line
Deformation
Stress (truss)
Line
Deformation
Mode Shape (truss)
Export
Reports
Settings
Properties
2D Plot Group
Label: Mode Shape (truss)
Data
Data set: Frecuencias Naturales/Solution 2
Eigenfrequency (Hz): 604.7
Title
Plot Settings
View: Automatic
x-axis label:
y-axis label:
Show hidden entities
Propagate hiding to lower dimensions
Plot data set edges
Color: Black
Frame: Material (X, Y, Z)
Color Legend
Show legends
Graphics
Convergence Plot 1
Eigenfrequency=604.7 Hz Line: Total displacement (ft)
ft
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
941 MB | 1044 MB

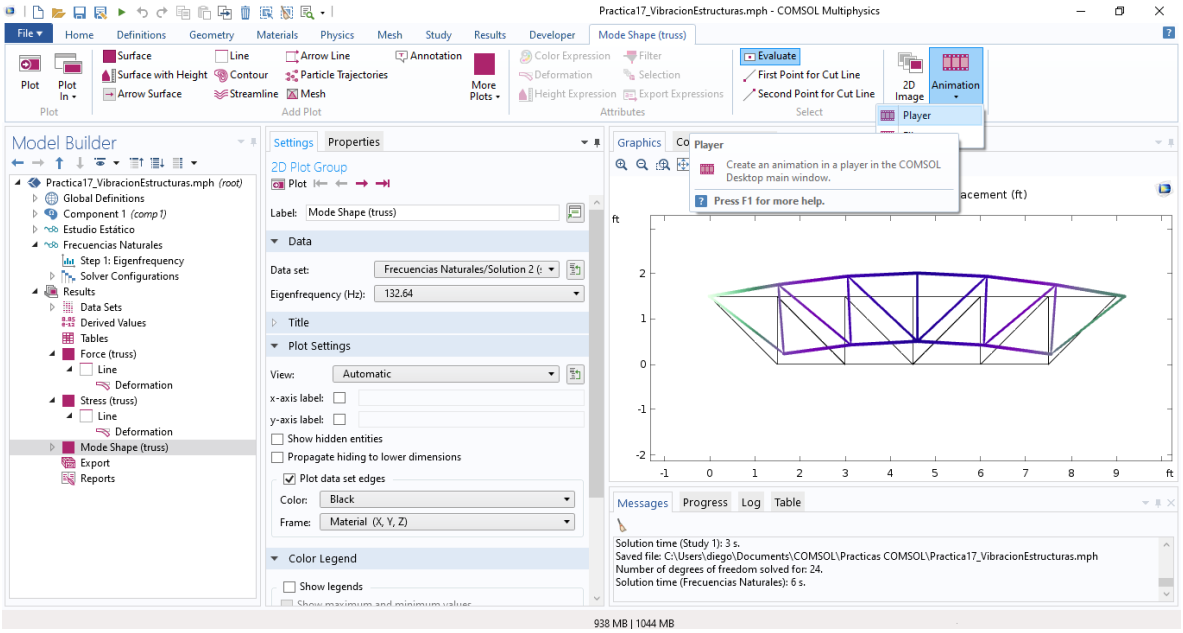


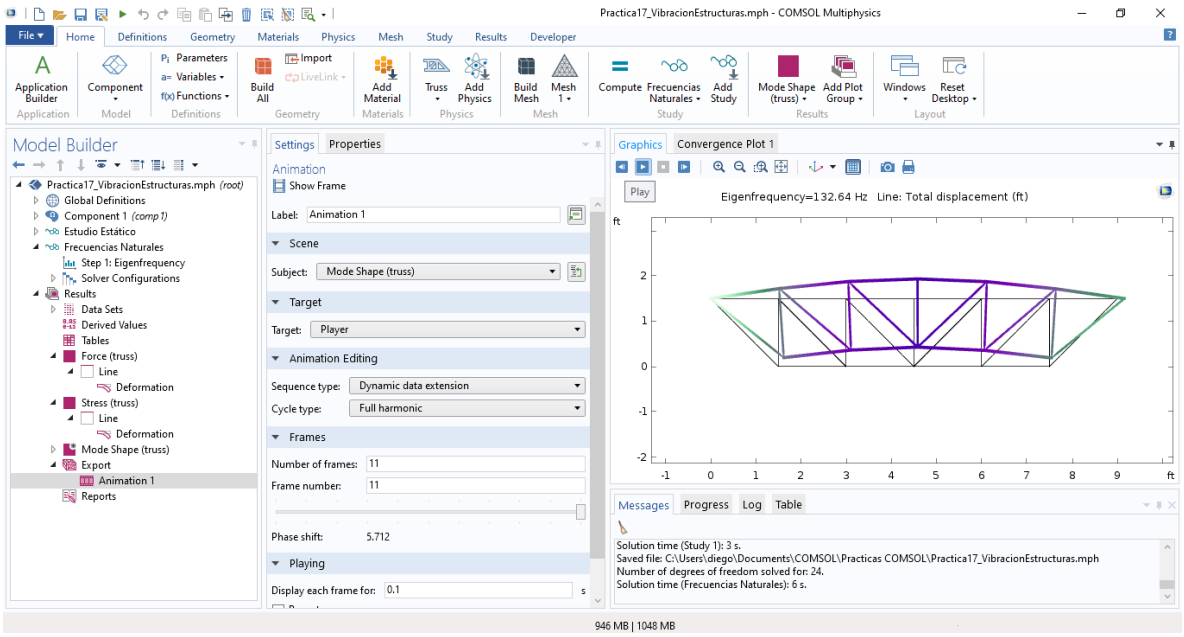


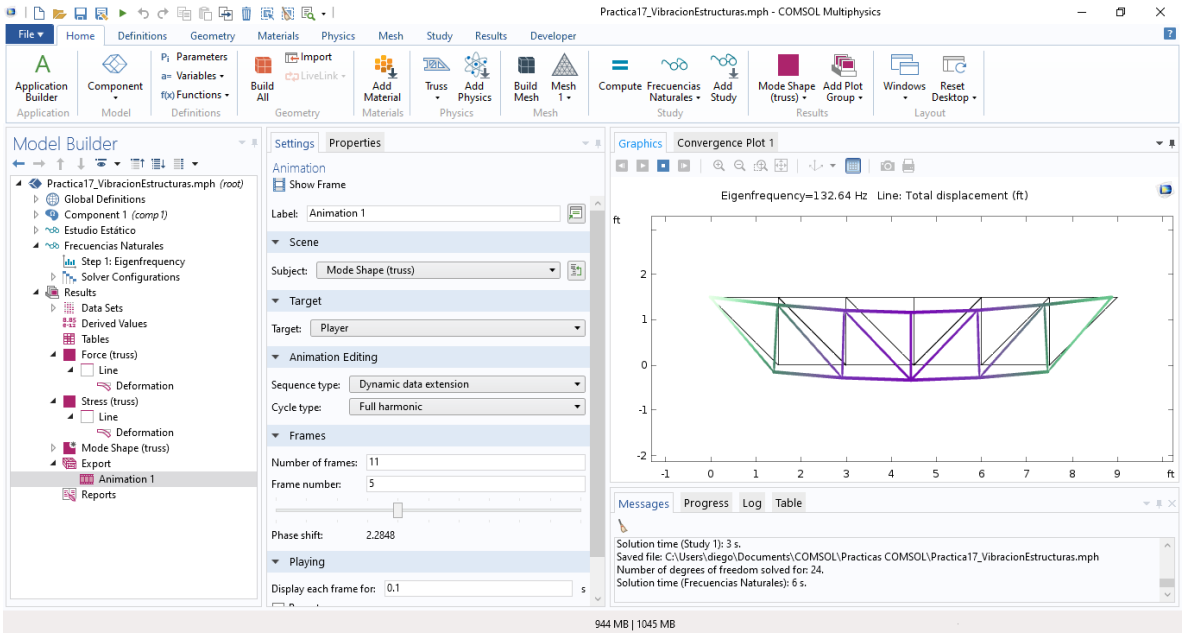
Se pueden observar animaciones del comportamiento de todas las vibraciones encontradas.



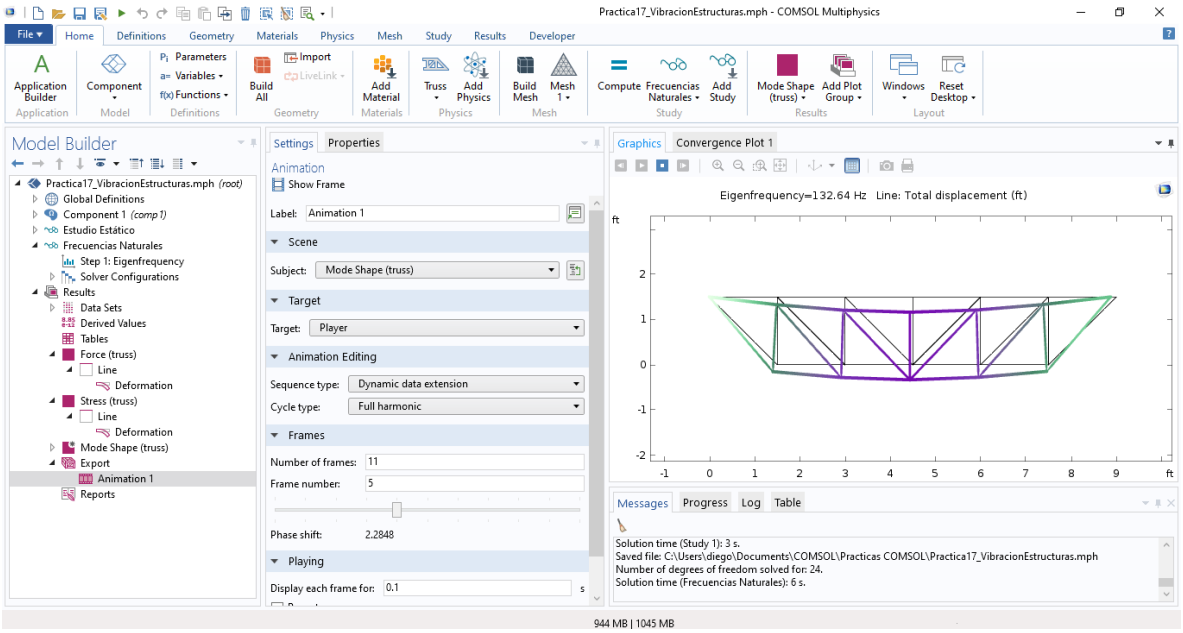
Con la opción de Player solo le doy play a la animación y con la opción de File se guarda la animación.

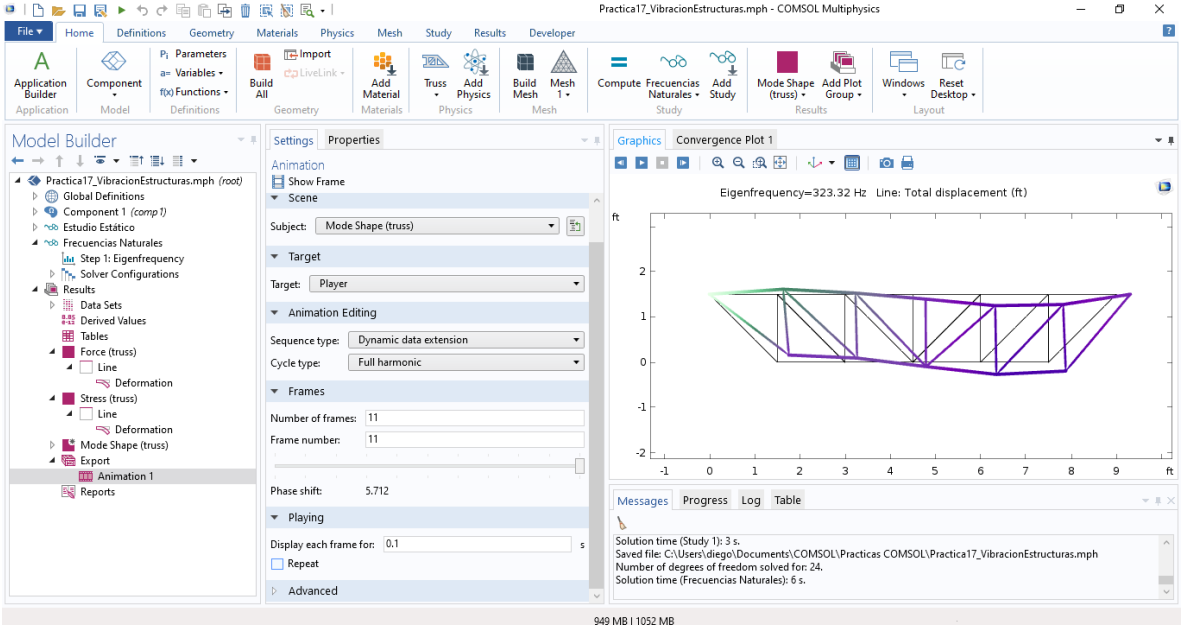


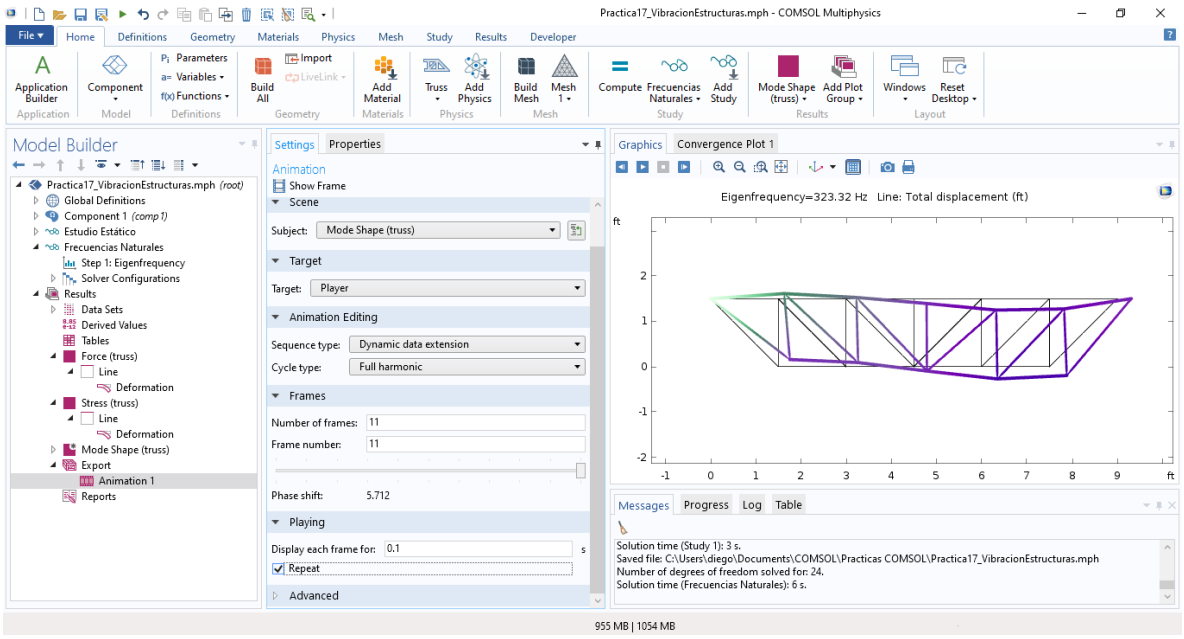




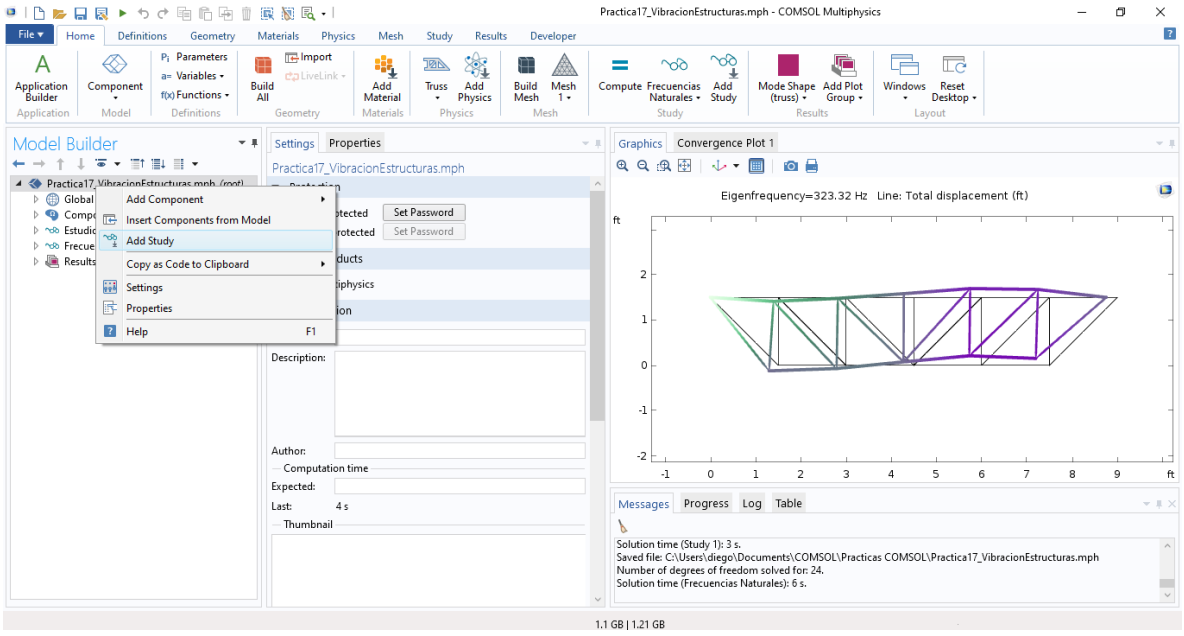
Dentro de la opción de Mode Shape es donde elijo la frecuencia que quiero ver.

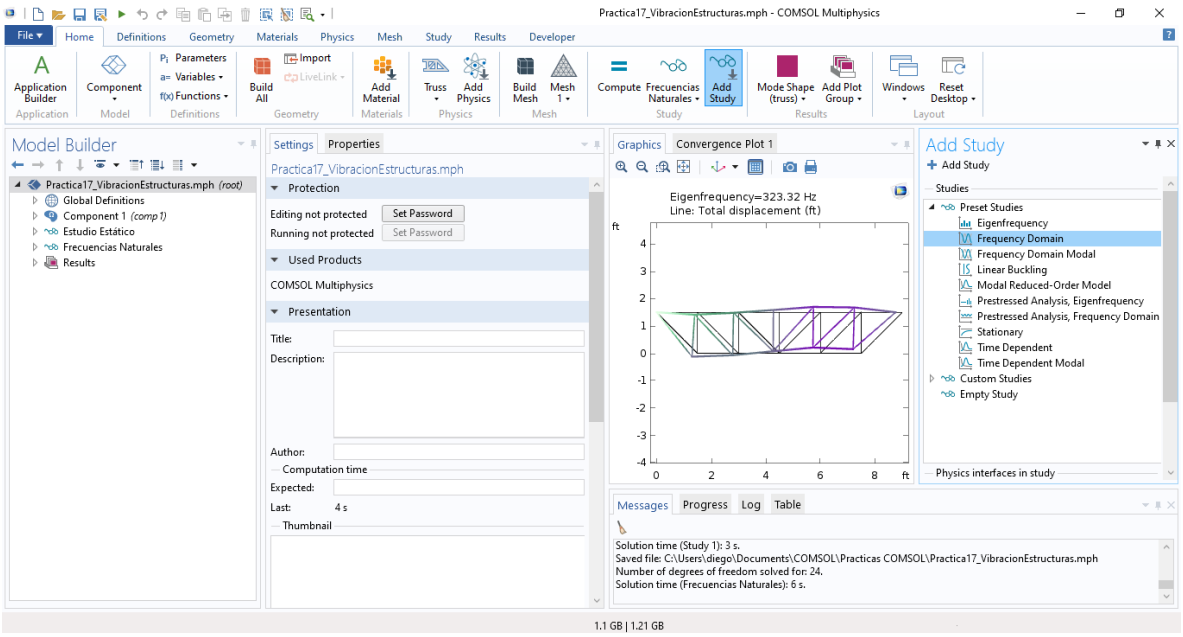

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Step 1: Eigenfrequency
Solver Configurations
Results
Data Sets
Derived Values
Tables
Force (truss)
Line
Deformation
Stress (truss)
Line
Deformation
Mode Shape (truss)
Export
Animation 1
Reports
Settings
Properties
Animation
Show Frame
Label: Animation 1
Scene
Subject: Mode Shape (truss)
Target
Target: Player
Animation Editing
Sequence type: Dynamic data extension
Cycle type: Full harmonic
Frames
Number of frames: 11
Frame number: 5
Phase shift: 2.2848
Playing
Display each frame for: 0.1 s
Graphics
Convergence Plot 1
Eigenfrequency=132.64 Hz Line: Total displacement (ft)
ft
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
944 MB | 1045 MB

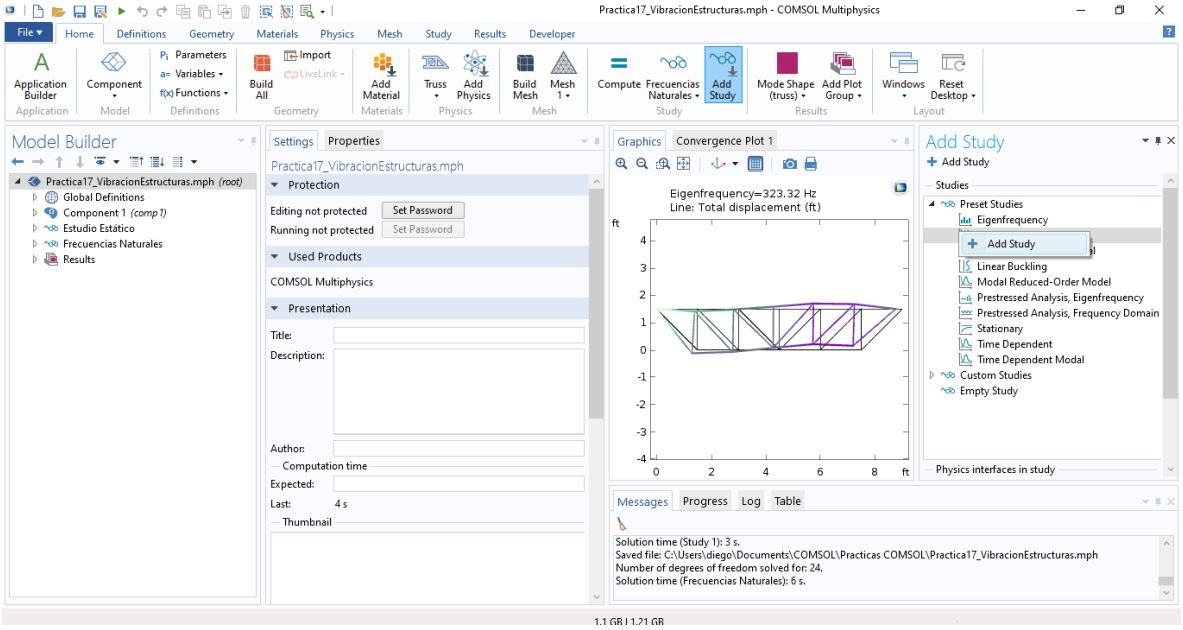

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Step 1: Eigenfrequency
Solver Configurations
Results
Data Sets
Derived Values
Tables
Force (truss)
Line
Deformation
Stress (truss)
Line
Deformation
Mode Shape (truss)
Export
Animation 1
Reports
Settings
Properties
Animation
Show Frame
Scene
Subject: Mode Shape (truss)
Target
Target: Player
Animation Editing
Sequence type: Dynamic data extension
Cycle type: Full harmonic
Frames
Number of frames: 11
Frame number: 11
Phase shift: 5.712
Playing
Display each frame for: 0.1 s
Repeat
Advanced
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz Line: Total displacement (ft)
ft
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
949 MB | 1052 MB

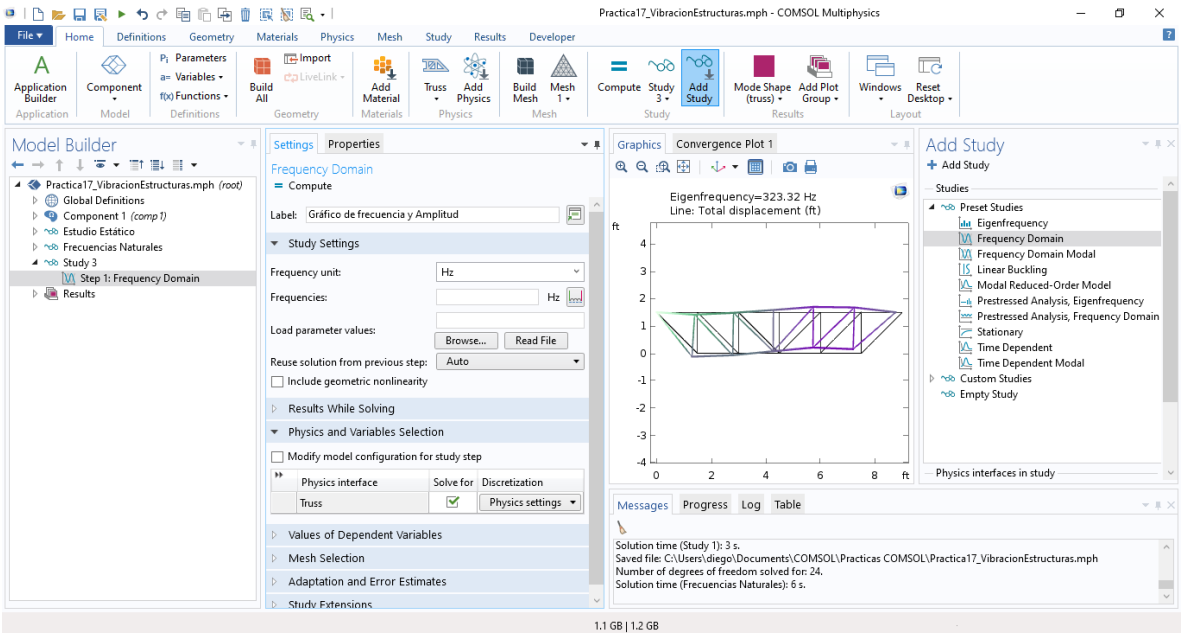


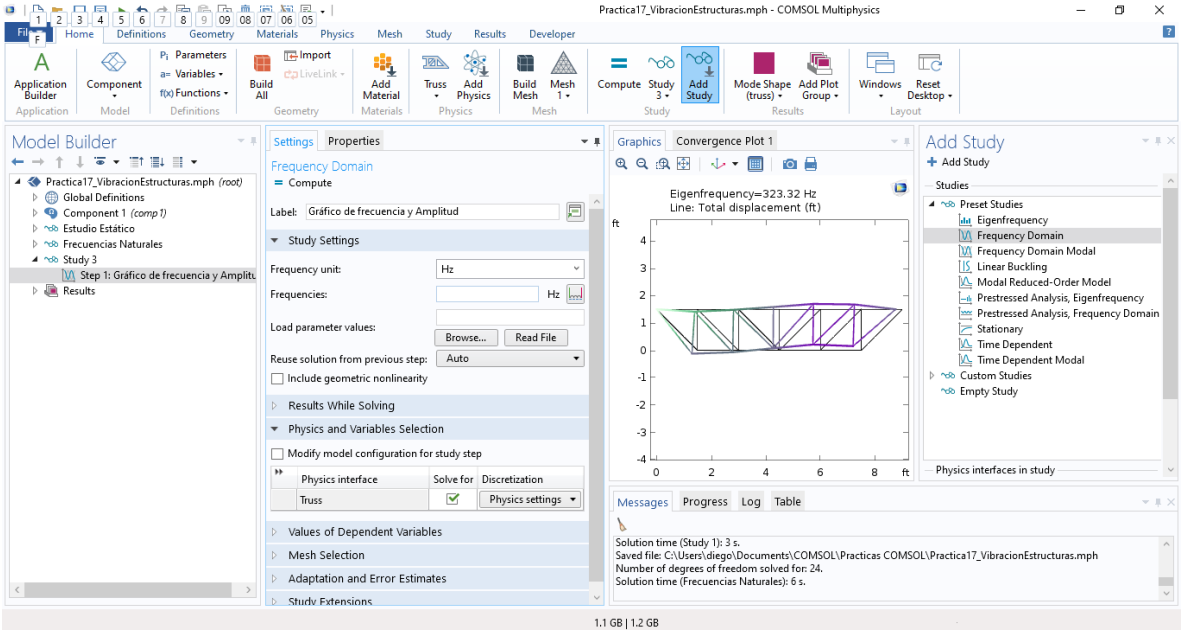
Toda la animación se puede poner dentro de una gráfica, pero para esto debo agregar otro estudio.



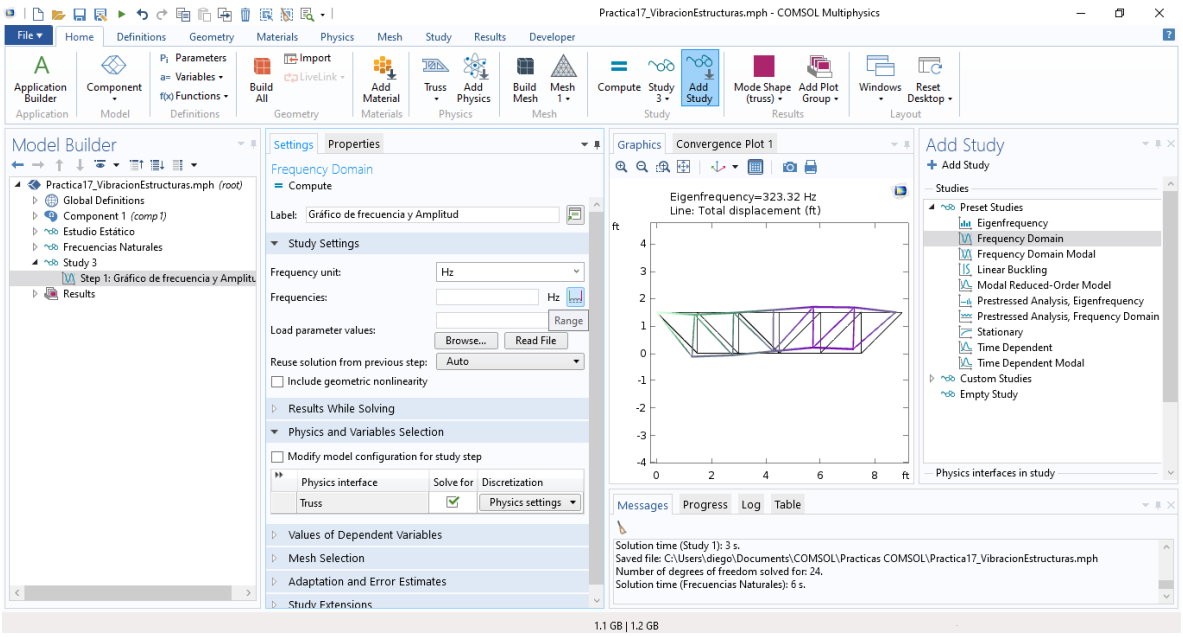

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Frecuencias Naturales
Add Study
Mode Shape (truss)
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Results
Settings
Properties
Practica17_VibracionEstructuras.mph
Protection
Editing not protected
Set Password
Running not protected
Set Password
Used Products
COMSOL Multiphysics
Presentation
Title:
Description:
Author:
Computation time
Expected:
Last: 4 s
Thumbnail
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz
Line: Total displacement (ft)
ft
Add Study
Add Study
Studies
Preset Studies
Eigenfrequency
Frequency Domain
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
1.1 GB | 1.21 GB

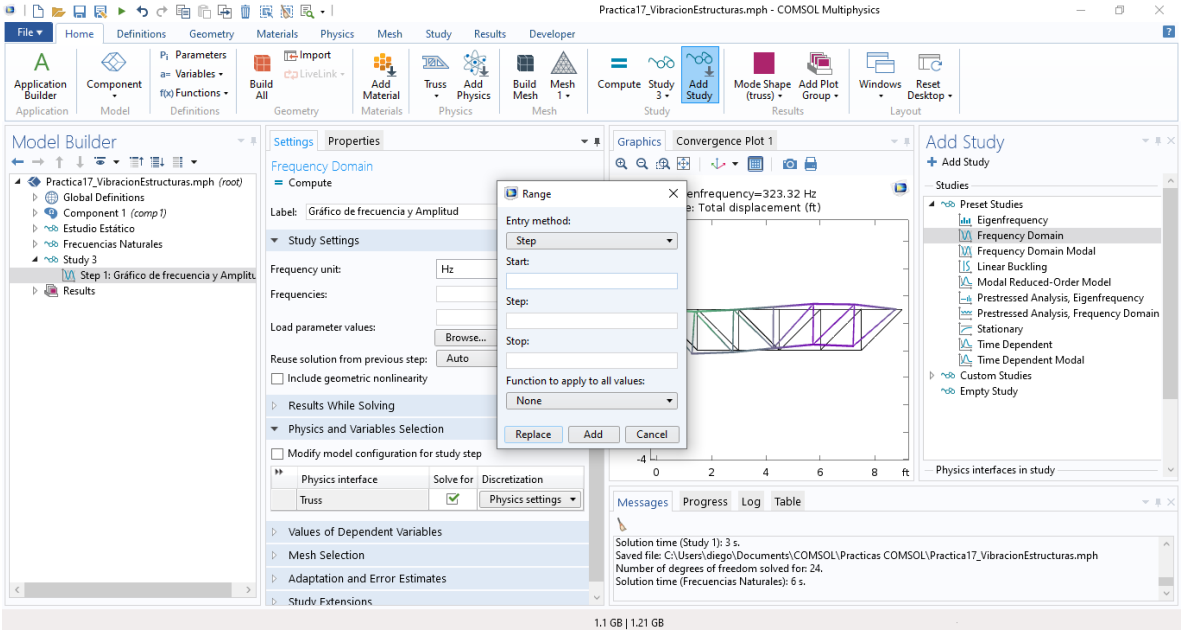

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Truss
Add Physics
Build Mesh
Mesh 1
Compute
Frecuencias Naturales
Add Study
Mode Shape (truss)
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Results
Settings
Properties
Practica17_VibracionEstructuras.mph
Protection
Editing not protected
Set Password
Running not protected
Set Password
Used Products
COMSOL Multiphysics
Presentation
Title:
Description:
Author:
Computation time
Expected:
Last: 4 s
Thumbnail
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz
Line: Total displacement (ft)
ft
Add Study
Add Study
Studies
Preset Studies
Eigenfrequency
Add Study
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
1.1 GB | 1.21 GB



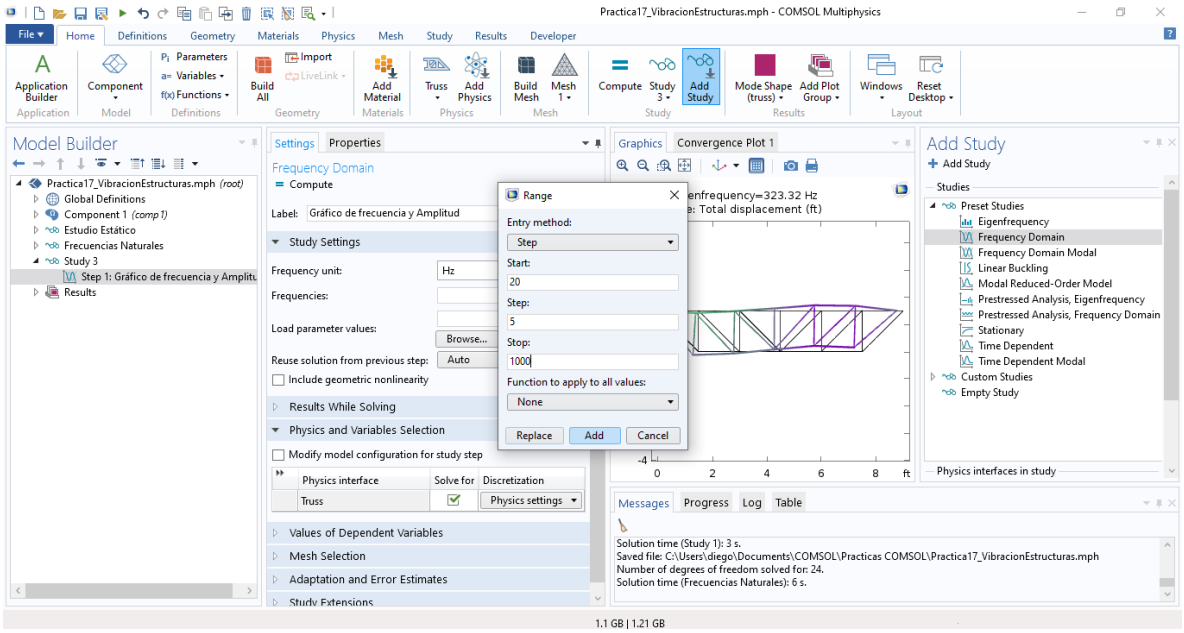


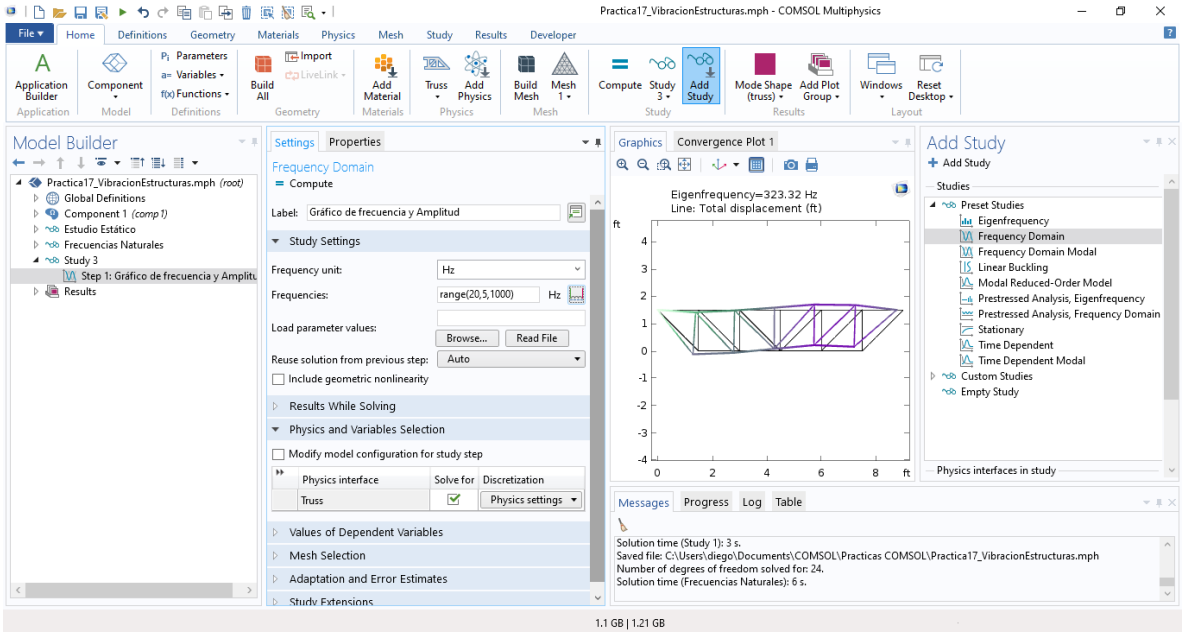
Donde dice frequencies debo meter el rango de las frecuencias que quiero graficar, esto es lo que irá en el eje horizontal de la gráfica resultante.

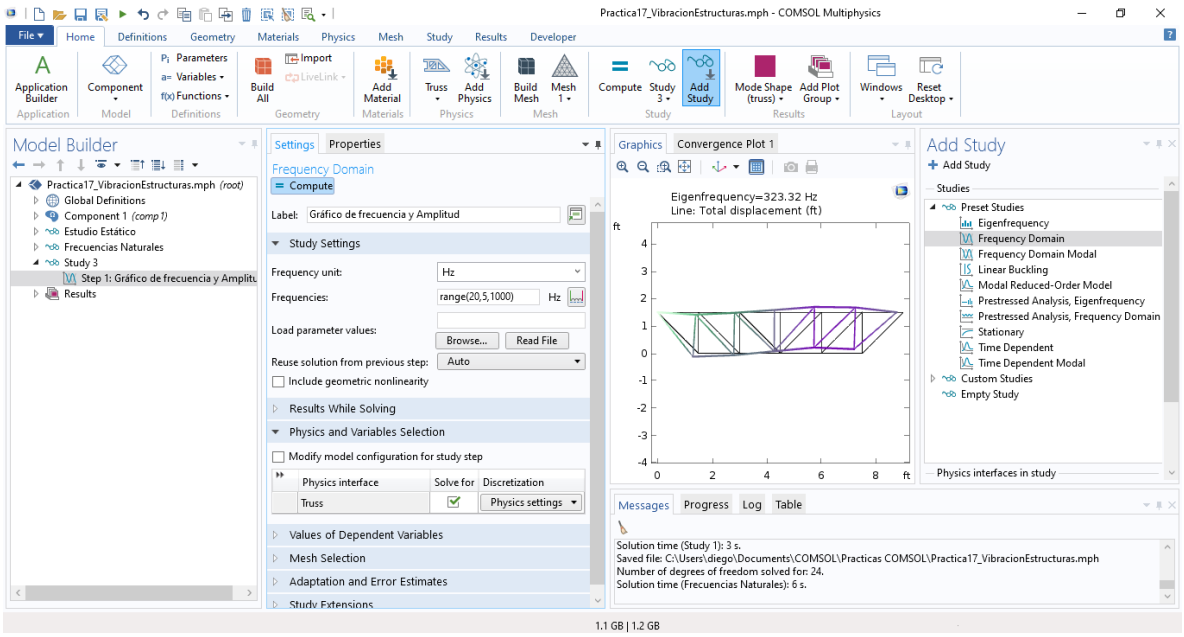


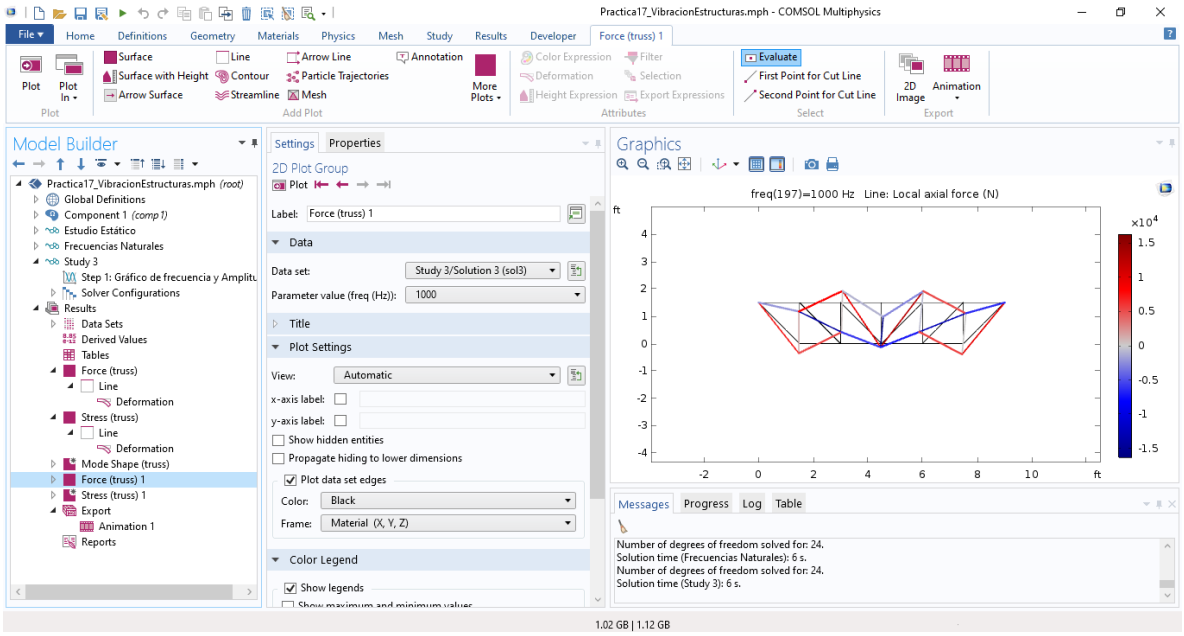


En esta ventana emergente le pongo su valor inicial y final y la separación entre ellos.

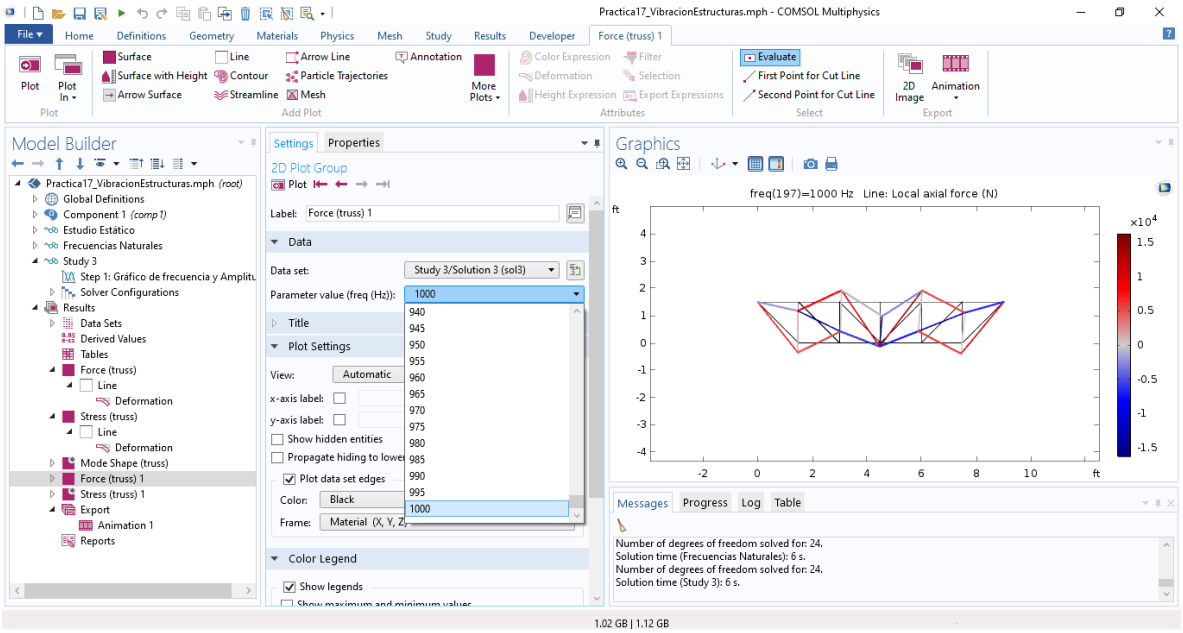

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Study 3
Step 1: Gráfico de frecuencia y Amplitu
Results
Settings
Properties
Frequency Domain
Compute
Label: Gráfico de frecuencia y Amplitud
Study Settings
Frequency unit: Hz
Frequencies:
Load parameter values:
Browse...
Reuse solution from previous step: Auto
Include geometric nonlinearity
Results While Solving
Physics and Variables Selection
Modify model configuration for study step
Physics interface
Solve for
Discretization
Truss
Physics settings
Values of Dependent Variables
Mesh Selection
Adaptation and Error Estimates
Study Extensions
Range
Entry method:
Step
Start:
20
Step:
5
Stop:
1000
Function to apply to all values:
None
Replace
Add
Cancel
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz
Line: Total displacement (ft)
Add Study
Studies
Preset Studies
Eigenfrequency
Frequency Domain
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
1.1 GB | 1.21 GB

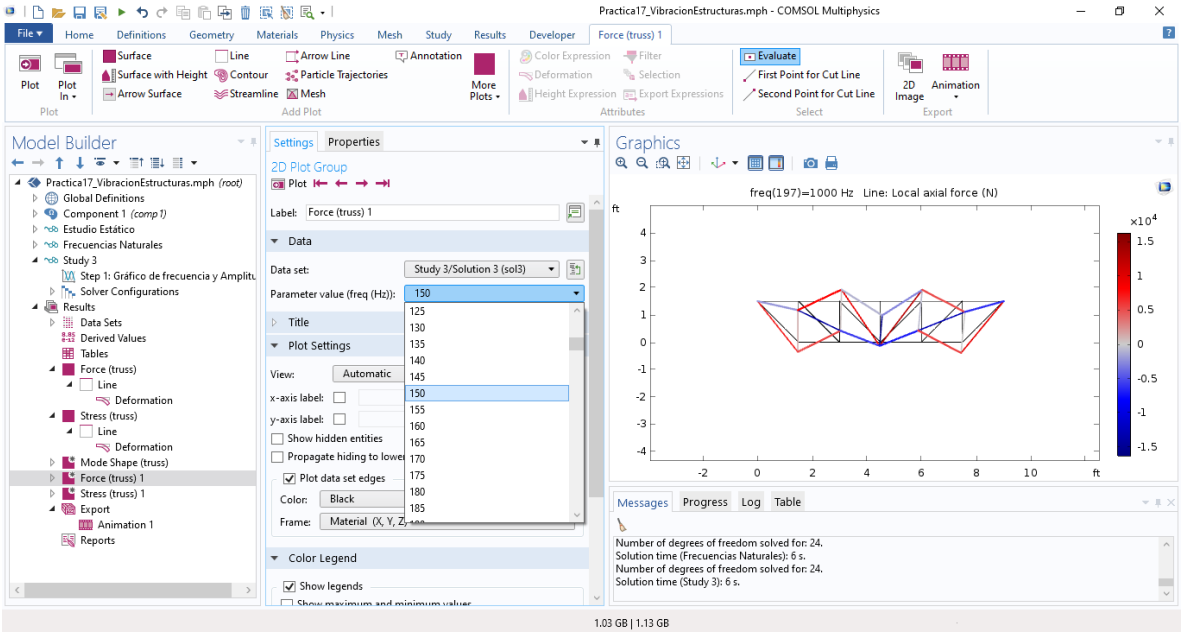

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Practica17_VibracionEstructuras.mph (root)
Global Definitions
Component 1 (comp 1)
Estudio Estático
Frecuencias Naturales
Study 3
Step 1: Gráfico de frecuencia y Amplitu
Results
Settings
Properties
Frequency Domain
Compute
Label: Gráfico de frecuencia y Amplitud
Study Settings
Frequency unit: Hz
Frequencies: range(20,5,1000) Hz
Load parameter values:
Browse...
Read File
Reuse solution from previous step: Auto
Include geometric nonlinearity
Results While Solving
Physics and Variables Selection
Modify model configuration for study step
Physics interface
Solve for
Discretization
Truss
Physics settings
Values of Dependent Variables
Mesh Selection
Adaptation and Error Estimates
Study Extensions
Graphics
Convergence Plot 1
Eigenfrequency=323.32 Hz
Line: Total displacement (ft)
ft
Add Study
Studies
Preset Studies
Eigenfrequency
Frequency Domain
Frequency Domain Modal
Linear Buckling
Modal Reduced-Order Model
Prestressed Analysis, Eigenfrequency
Prestressed Analysis, Frequency Domain
Stationary
Time Dependent
Time Dependent Modal
Custom Studies
Empty Study
Physics interfaces in study
Messages
Progress
Log
Table
Solution time (Study 1): 3 s.
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica17_VibracionEstructuras.mph
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
1.1 GB | 1.21 GB

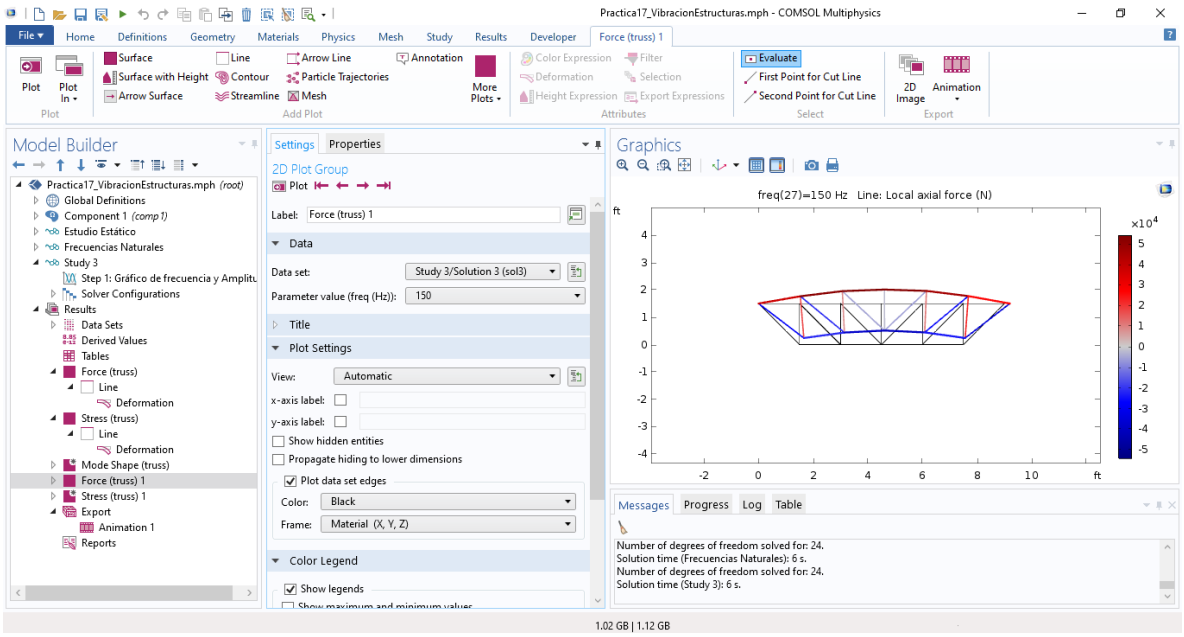




Aquí me mostrará que le pasará para todas las frecuencias que le indique.

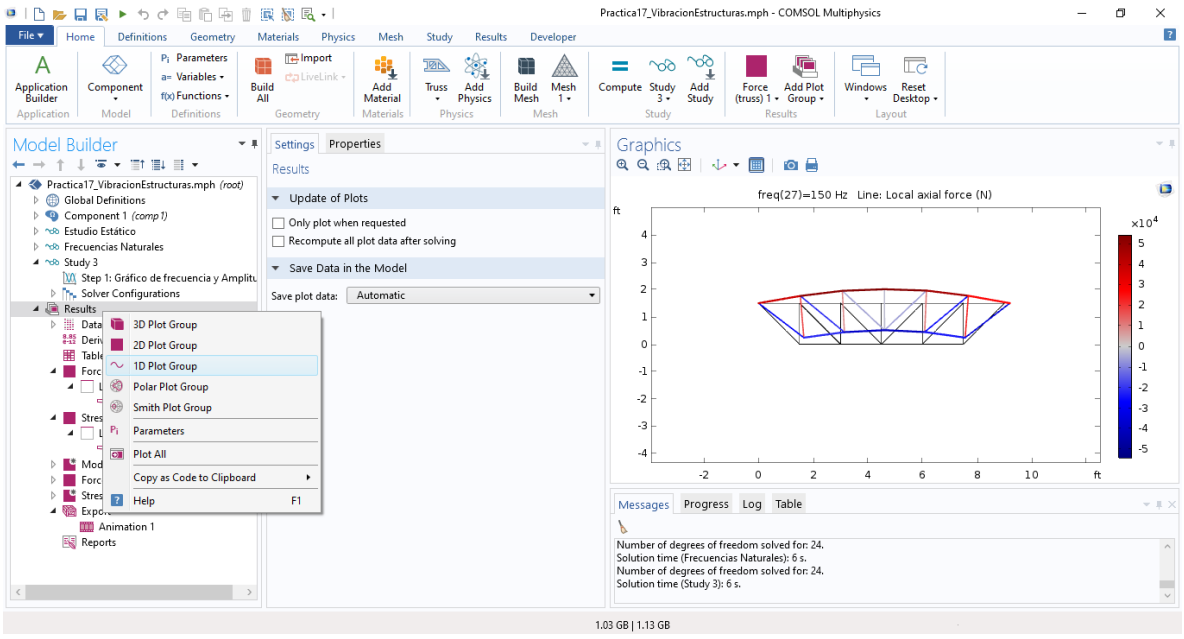

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Label: Force (truss) 1
Data
Data set: Study 3/Solution 3 (sol3)
Parameter value (freq (Hz)): 1000
940
945
950
955
960
965
970
975
980
985
990
995
1000
Title
Plot Settings
View: Automatic
x-axis label:
y-axis label:
Show hidden entities
Plot data set edges
Color: Black
Color Legend
Show legends
Graphics
freq(197)=1000 Hz Line: Local axial force (N)
Messages
Progress
Log
Table
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 3): 6 s.
1.02 GB | 1.12 GB

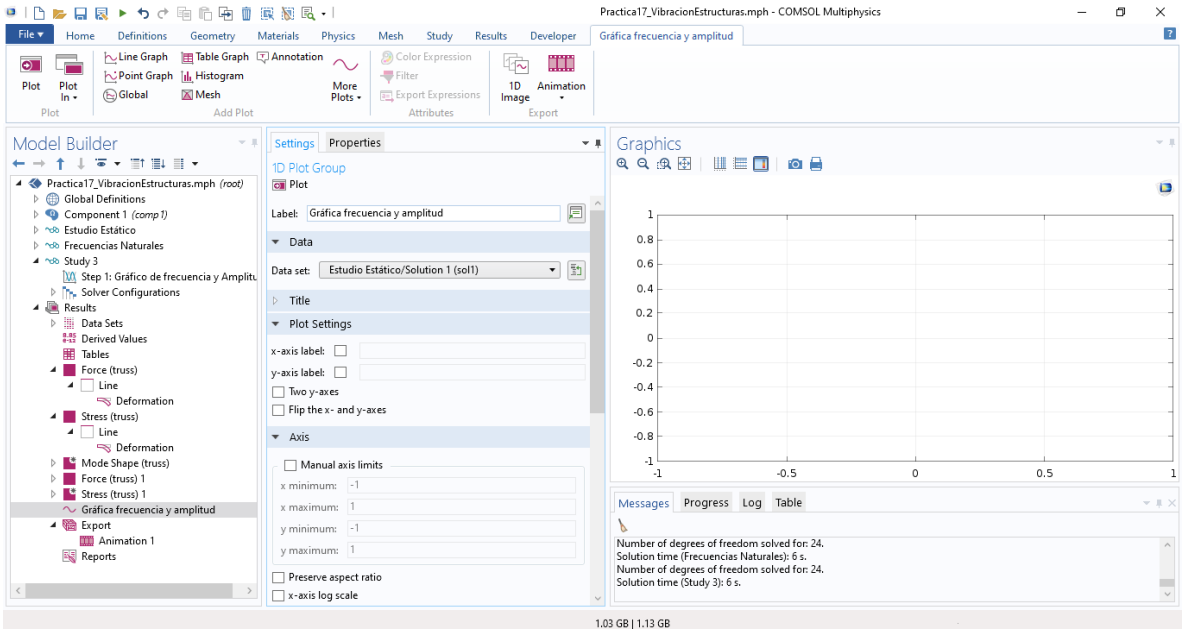

Practica17_VibracionEstructuras.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Label: Force (truss) 1
Data
Data set: Study 3/Solution 3 (sol3)
Parameter value (freq (Hz)): 150
125
130
135
140
145
150
155
160
165
170
175
180
185
Title
Plot Settings
View: Automatic
x-axis label:
y-axis label:
Show hidden entities
Plot data set edges
Color: Black
Color Legend
Show legends
Graphics
freq(197)=1000 Hz Line: Local axial force (N)
Messages
Progress
Log
Table
Number of degrees of freedom solved for: 24.
Solution time (Frecuencias Naturales): 6 s.
Number of degrees of freedom solved for: 24.
Solution time (Study 3): 6 s.
1.03 GB | 1.13 GB



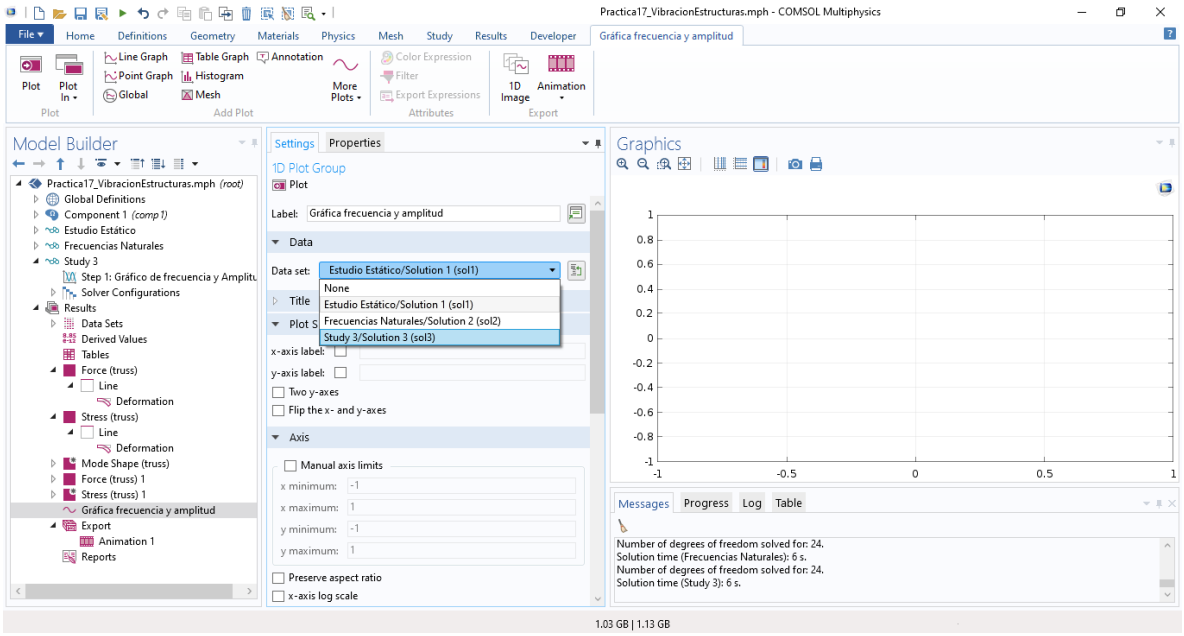
GRÁFICA DE FRECUENCIAS ARMÓNICAS:

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

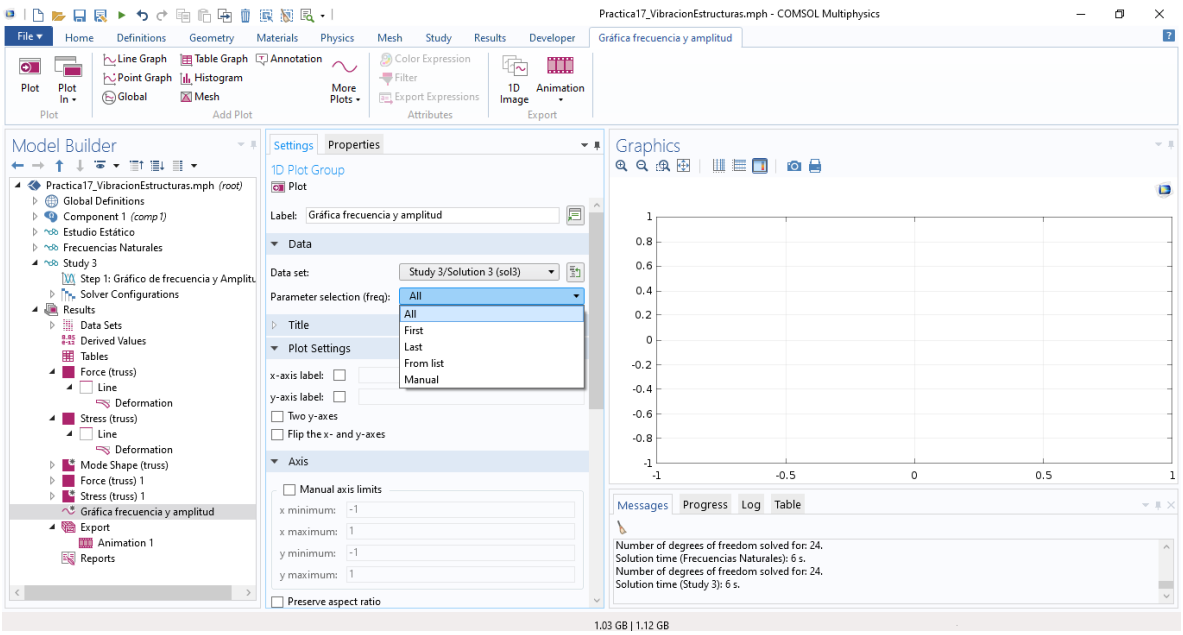


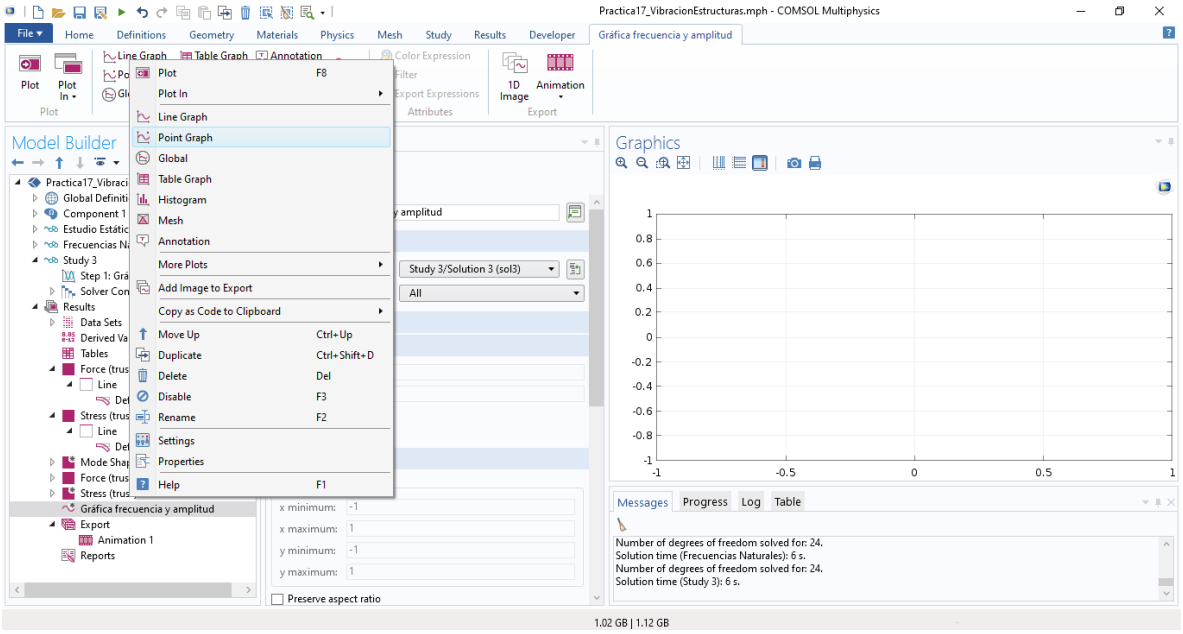


Luego selecciono el estudio que quiero graficar.

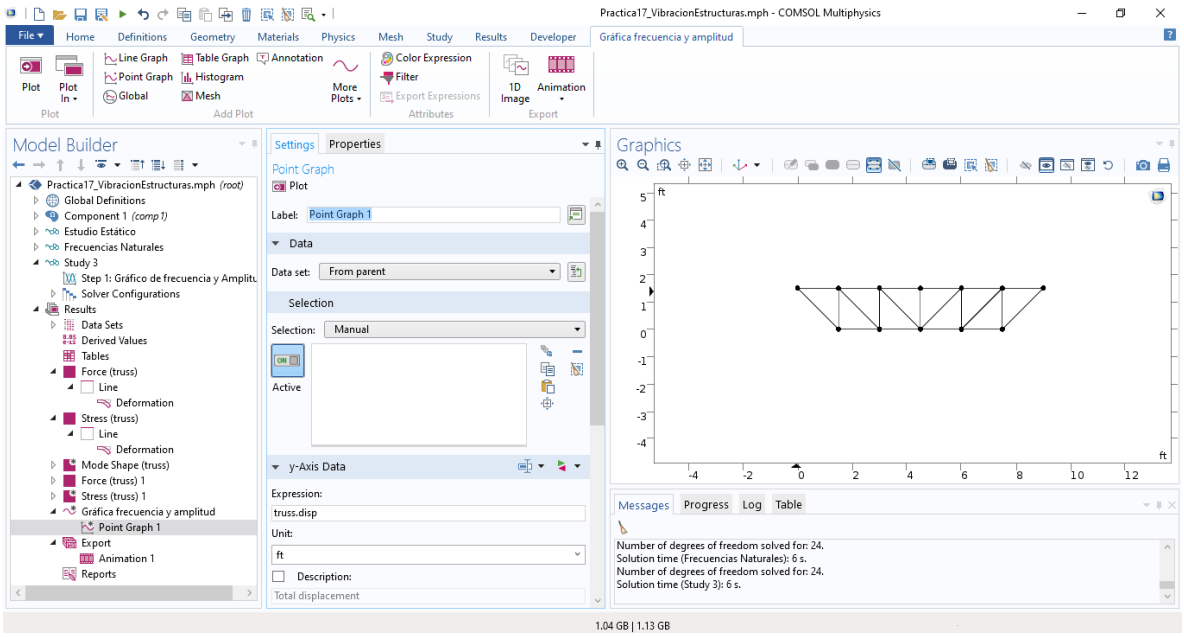


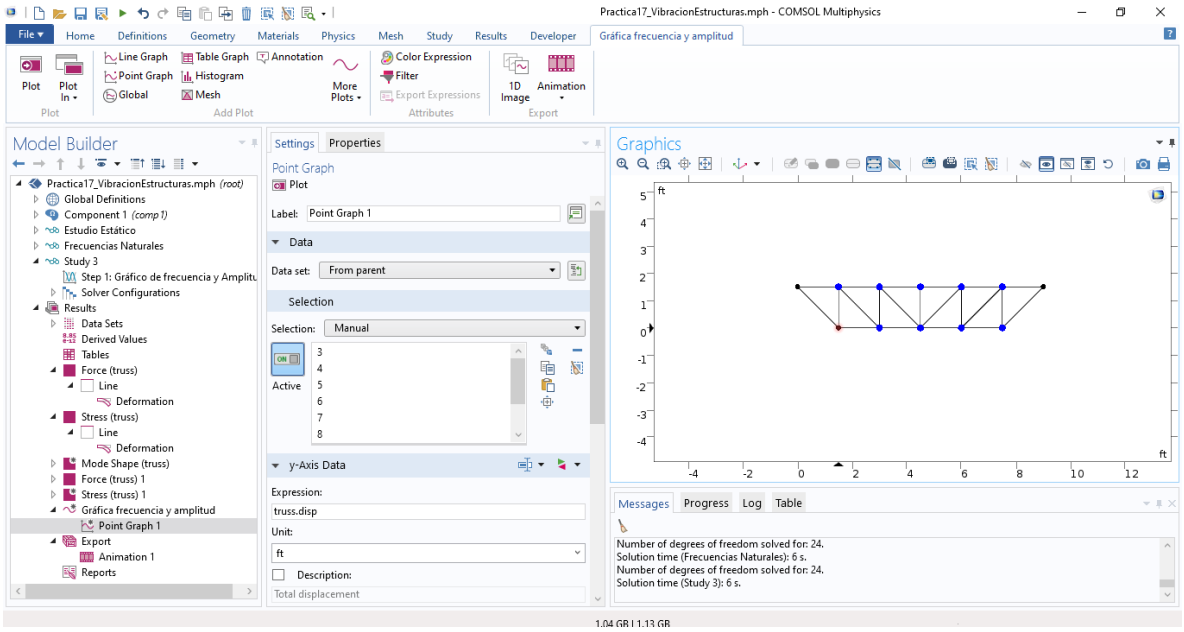
Después hay que indicar que resultados se van a poner en el gráfico.



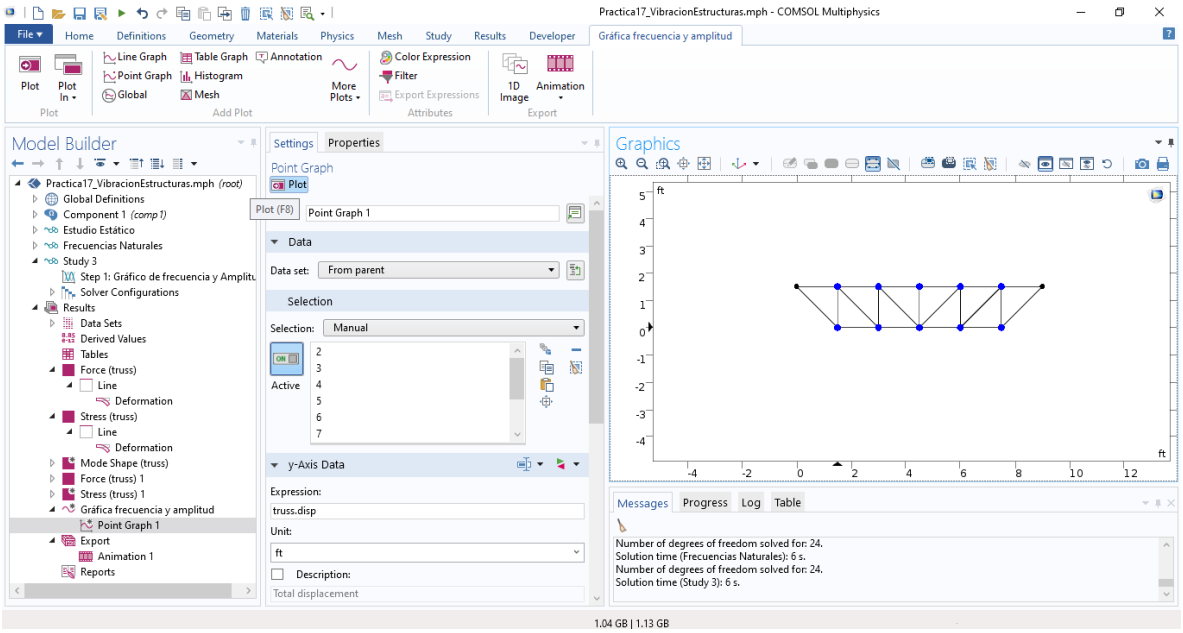


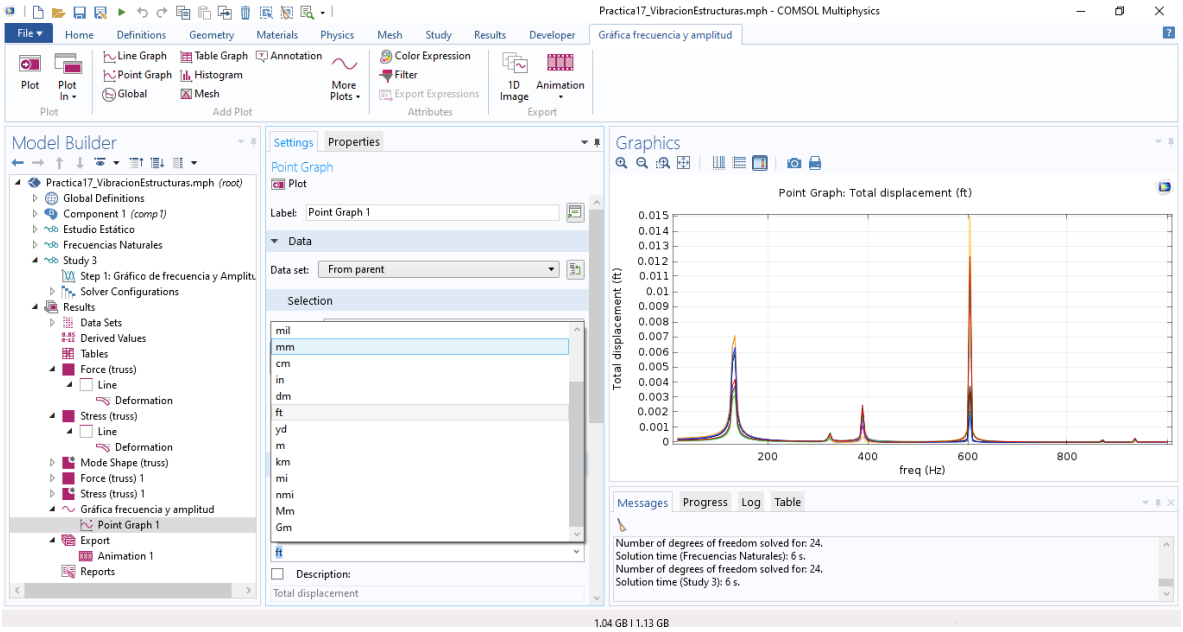
Luego debo seleccionar en qué punto quiero que se grafique la frecuencia natural contra la amplitud del movimiento de la estructura.





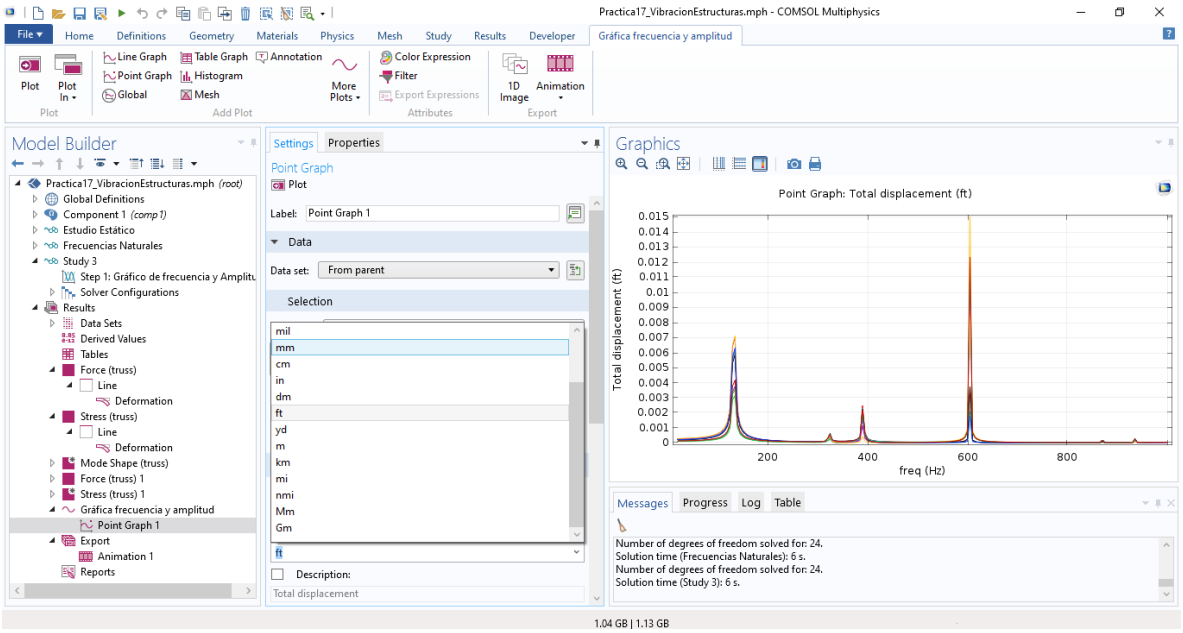
Puedo seleccionar uno, varios o todos los nodos.



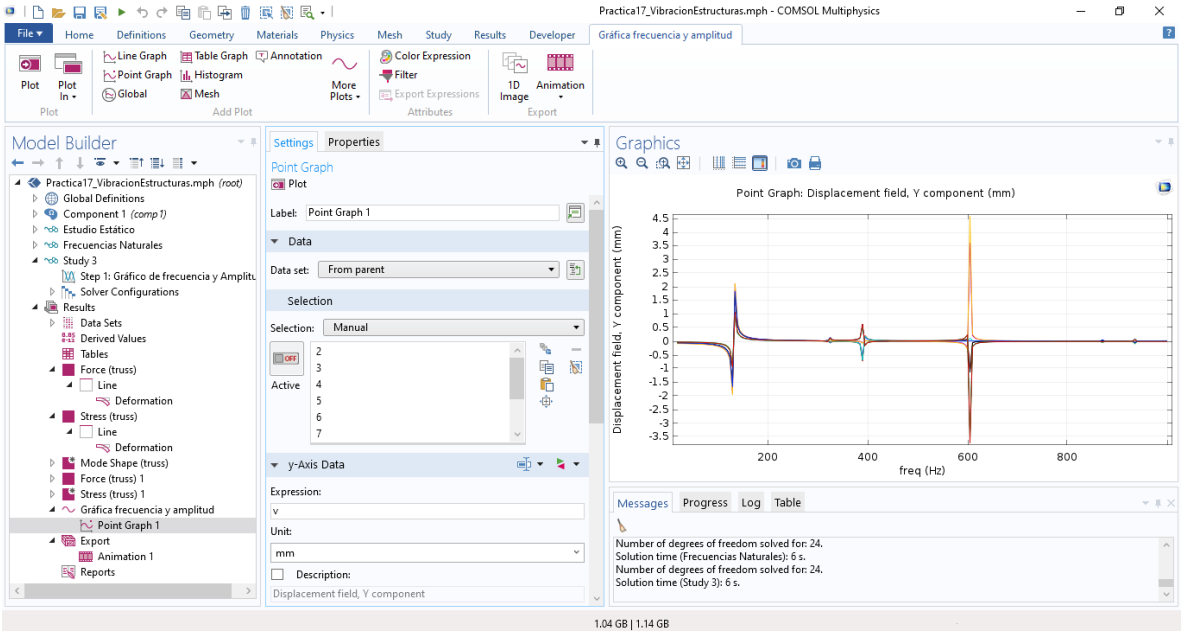


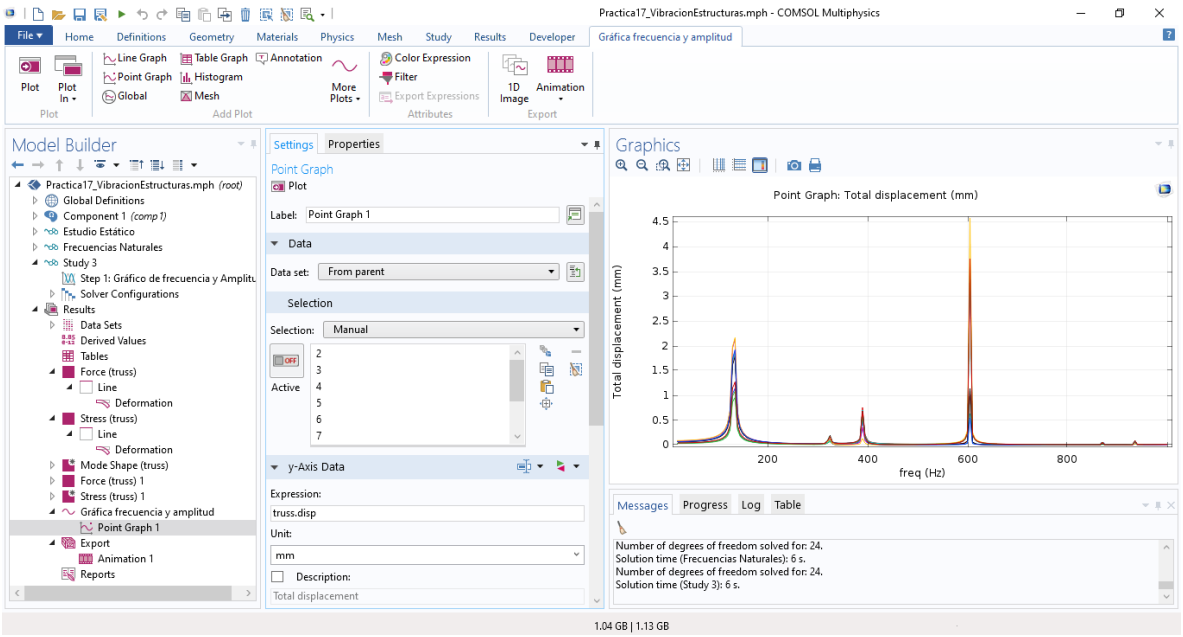
Es posible seleccionar la unidad del eje vertical que muestra el desplazamiento de los nodos en mi estructura.

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



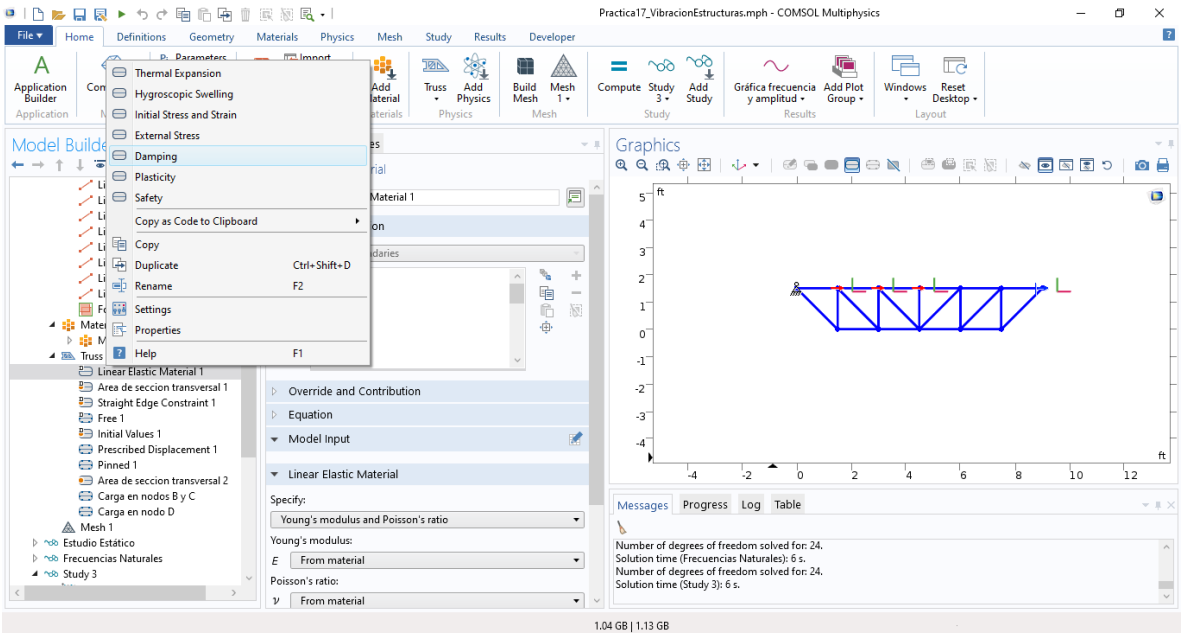
Con la instrucción ***truss.disp*** lo que se grafica es el desplazamiento total, pero puedo poner igual v o u para que se vea el desplazamiento vertical u horizontal, esto se verá en forma asintótica porque el desplazamiento puede ser positivo y negativo hacia el infinito.



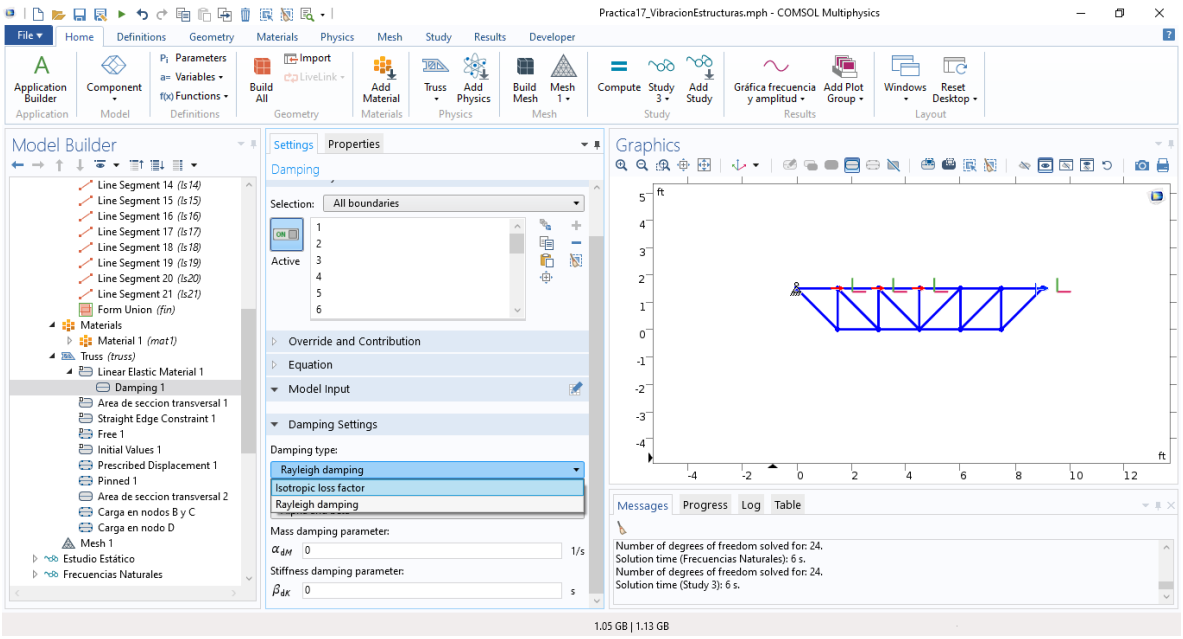


En la vida real esto no pasa tan exagerado porque el material tiene una propiedad de disipación de energía que funciona como un tipo amortiguador.

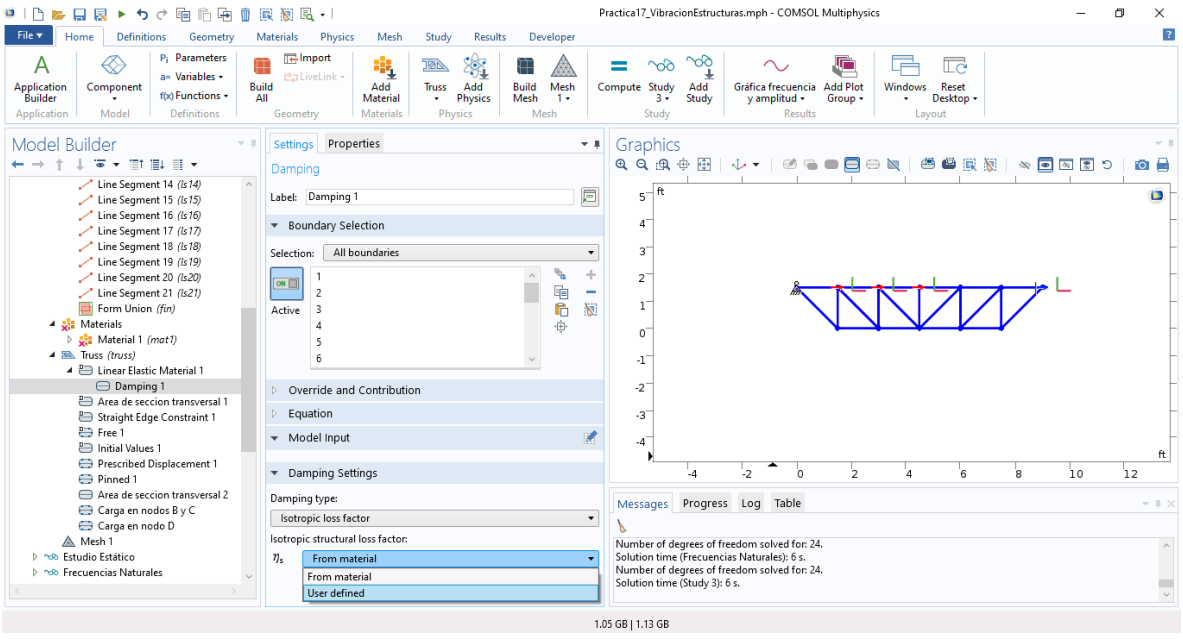
Esto se agrega en la estructura dentro de Truss con la instrucción Damping.



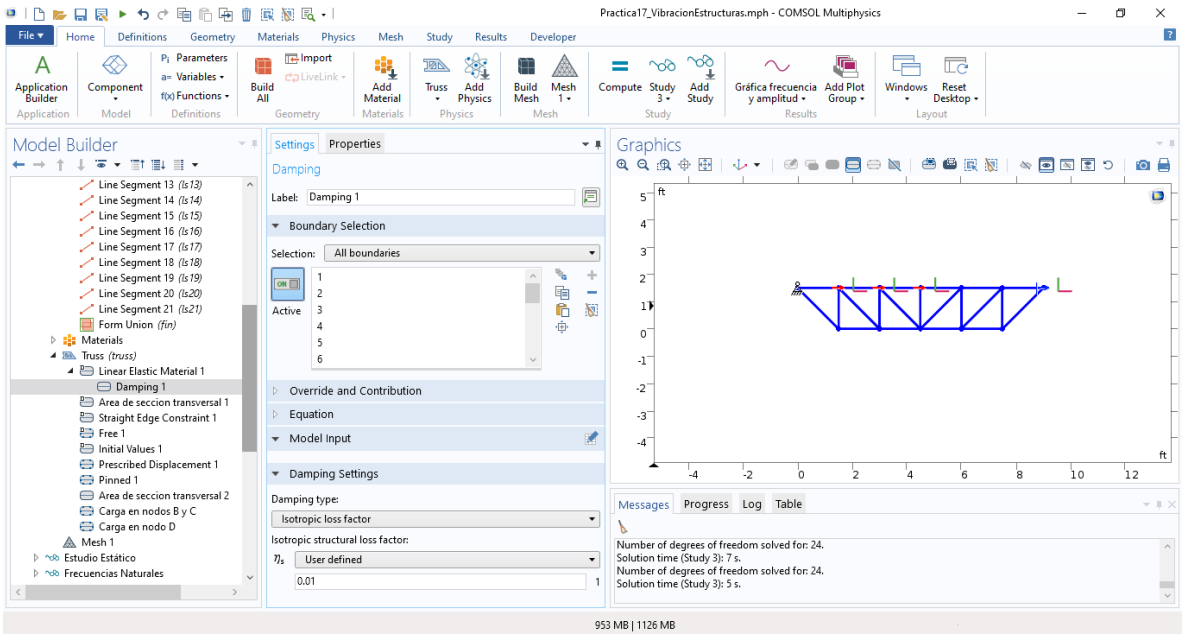
El amortiguamiento del material en el programa se mete como Isotropic loss factor.



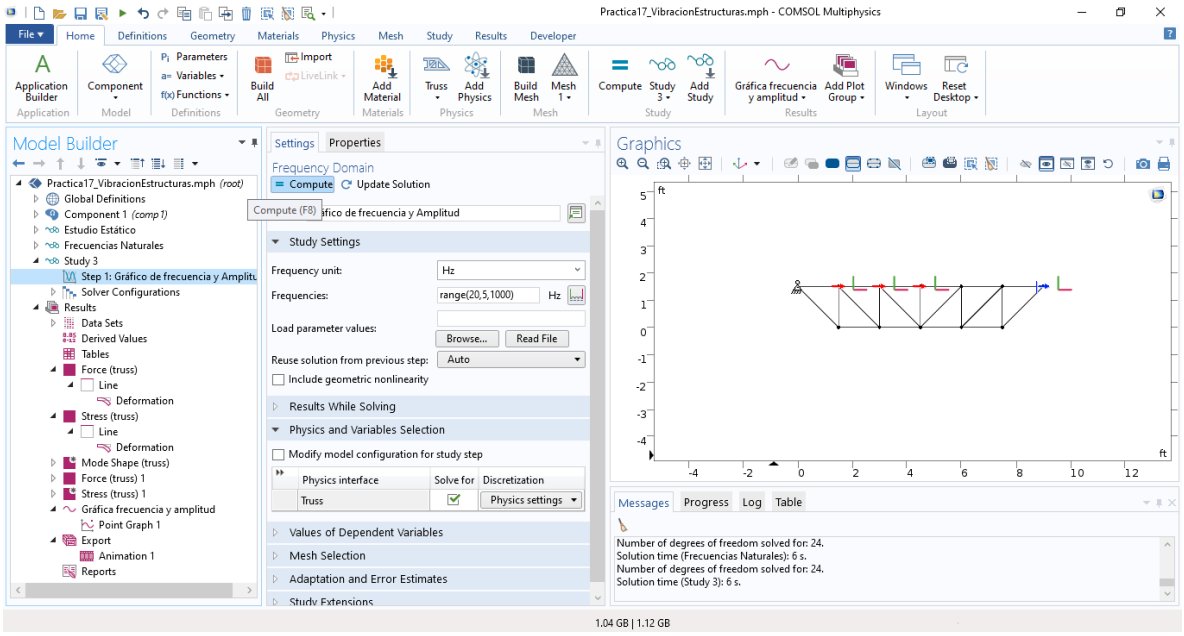
Este es isotrópico y lo que significa es que sus propiedades no cambiarán no importando el punto en el que lo analice, esto en la vida real pasa con el acero.

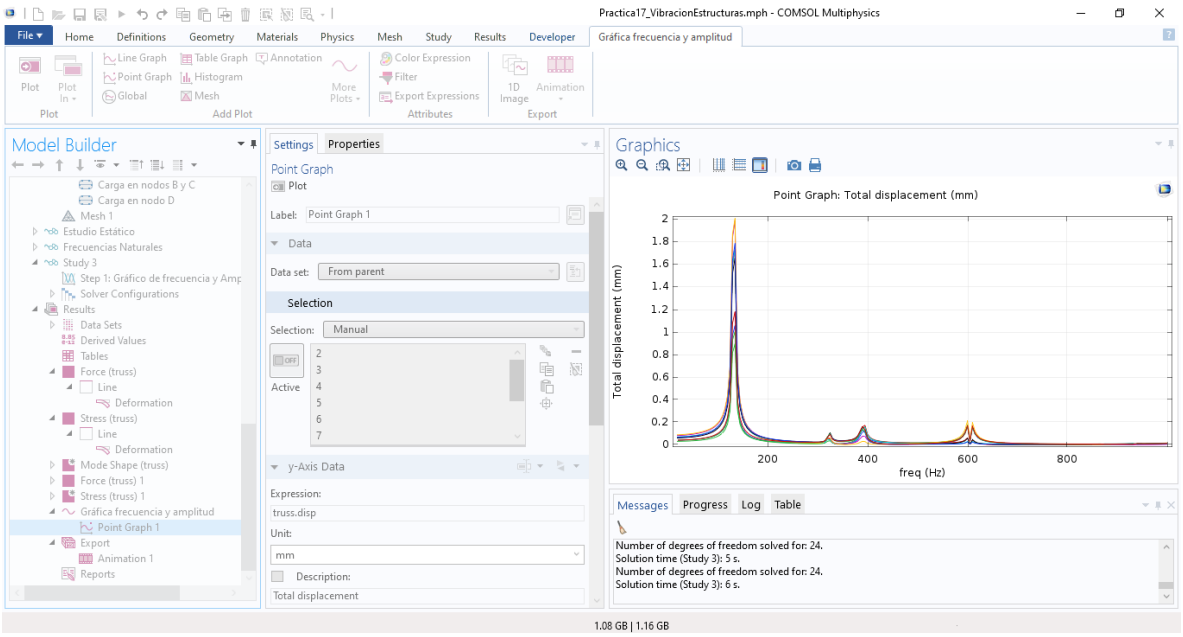


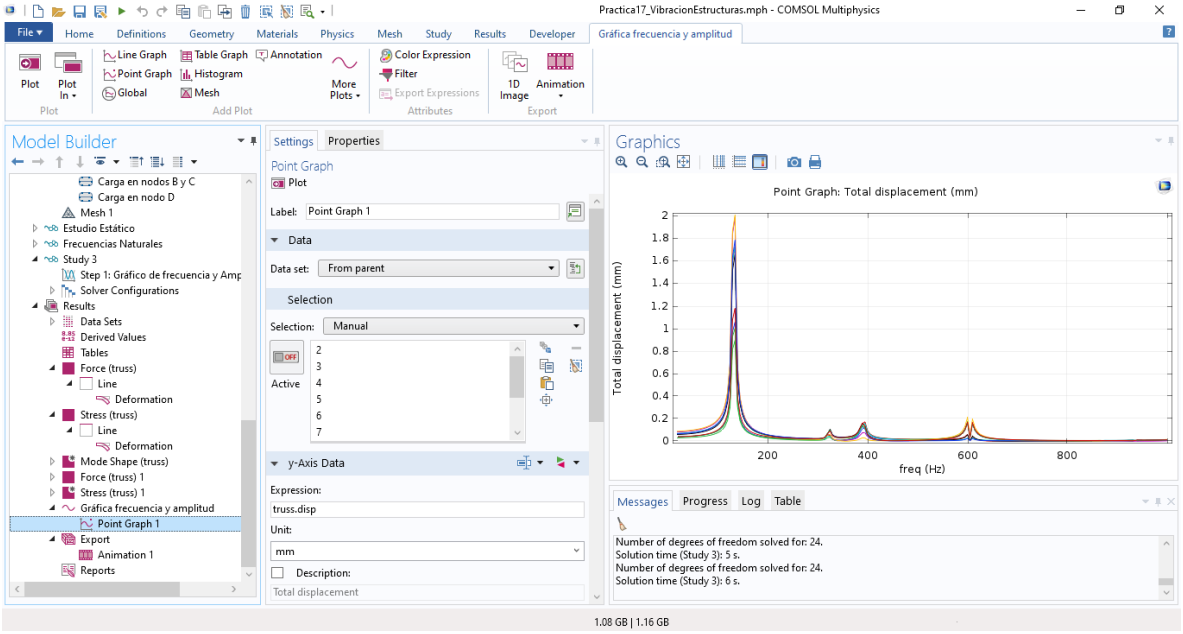
En el acero la amortiguación es de 0.01.



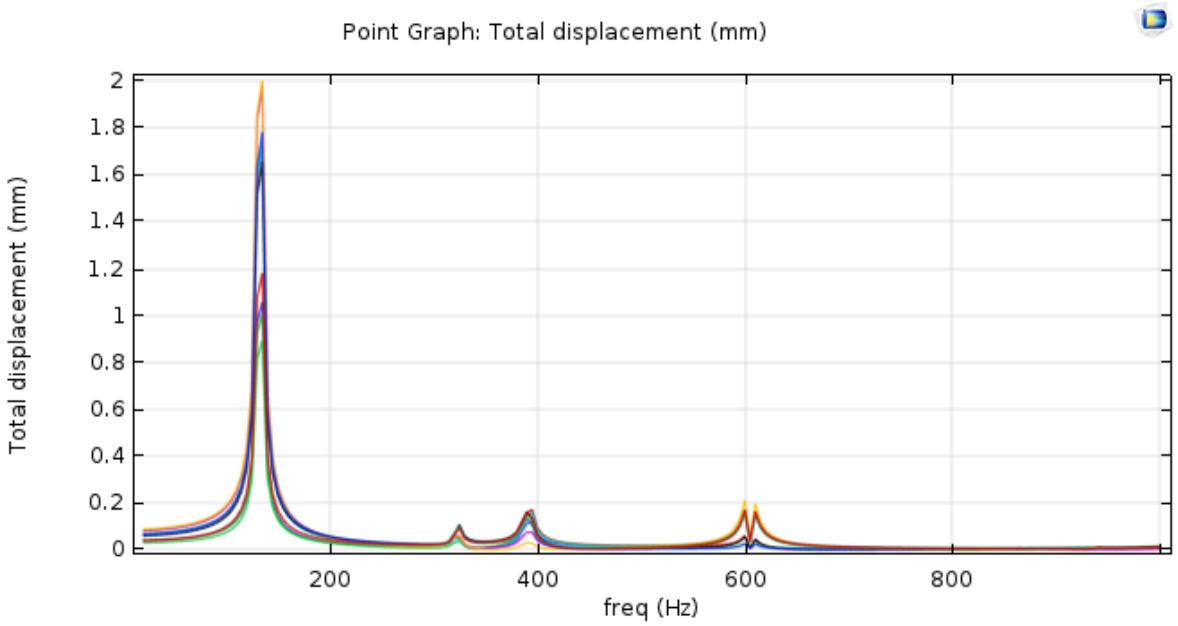
Y vuelvo a hacer el cálculo de el gráfico de frecuencia.







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.