INGENIERÍA MECATRÓNICA

DI_CERO

DIEGO CERVANTES RODRÍGUEZ

INGENIERÍA ASISTIDA POR COMPUTADORA

COMSOL MULTIPHYSICS 5.6

18: Transferencia Estacionaria de Calor

Contenido

INTRODUCCIÓN TEÓRICA:

Transferencia estacionaria de calor

Condiciones iniciales: A continuación, se estudia la transmisión de calor a través de dos capas de diferentes materiales. Si bien es cierto que se presentan dos mecanismos combinados de transferencia de calor, convección y conducción, se hace especial énfasis en la transferencia conductiva para así poder disponer de solución analítica con la que podremos contrastar los resultados obtenidos mediante el programa COMSOL Multiphysics.

El punto de partida es la ley de Fourier para la conducción de calor que nos permite expresar el **flujo de calor, *q***, en función de la conductividad térmica, *k*, y del gradiente de temperaturas, *dT/dx*.

$$q = -k\,\frac{dT}{dx}$$

En el caso de una placa plana de espesor *e* con flujo de calor constante, el flujo de calor se expresa en términos de la **potencia calorífica $\dot{Q}$** y la superficie de la placa (área que cruza el flujo de calor), *A*, como:

$$q = \frac{\dot{Q}}{A}$$

Lo cual permite escribir:

$$\frac{\dot{Q}}{A} = -k\,\frac{dT}{dx}$$

$$\dot{Q}dx = -k\,A\,dT$$

$$\int_0^e \dot{Q}dx = -k\,A\int_{T_1}^{T_2} dT$$

$$\dot{Q}e = -k\,A\,(T_2 - T_1)$$

Donde T_1 es la temperatura inicial (zona caliente) y T_2 la temperatura final (zona fría)

De manera que disponemos de expresiones para la potencia calorífica transmitida y para el flujo de calor dadas, respectivamente, por

$$\dot{Q} = k\,A\,\frac{(T_1 - T_2)}{e}$$

Las unidades serán W = J / s (watts = Joles / segundo).

$$q = k\,\frac{(T_1 - T_2)}{e}$$

Las unidades serán W / m^2 (watts / metro cuadrado).

A partir de aquí se trabaja con el coeficiente global de conducción del calor por unidad de superficie, definido por:

$$U_T = \frac{k}{e}$$

Con la resistencia térmica asociada que es

$$R_T = \frac{1}{U_T} = \frac{e}{k}$$

Con las ecuaciones de potencia y flujo, tendremos

$$\dot{Q} = (U_T\ e)\ A\ \frac{(T_1 - T_2)}{e} = U_T\ A\ (T_1 - T_2)$$

$$q = (U_T\ e)\ \ \frac{(T_1 - T_2)}{e} = U_T\ (T_1 - T_2)$$

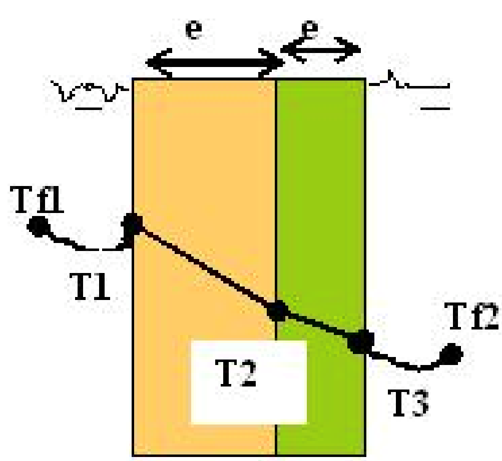
Tomando la convección de calor entre el aire circundante a la placa y la pared, modelaremos con un coeficiente de convección, *h*, es decir, con una resistencia térmica relacionada, *1/h*.

Finalmente, la resistencia térmica total de un material compuesto formado por distintas capas planas superpuestas (de espesores *e*) se puede estudiar a partir de la analogía de resistencias en serie:

$$R_T = \frac{1}{h_i} + \frac{e_1}{k_1} + \frac{e_2}{k_2} + \cdots + \frac{e_n}{k_n} + \frac{1}{h_e}$$

DESCRIPCIÓN DEL PROBLEMA:

Se propone estudiar la conducción del calor a través de un cuerpo plano constituido por placas planas, formado por dos capas superpuestas de espesores iguales de 10 cm. Se estudia un área de superficie de 1 m^2. El primer material tiene una conductividad térmica k_1 = 5 W/(m K) y el segundo material tiene una conductividad térmica k_2 = 0.4 W/(m K). El coeficiente de conductividad con el entorno *h* es de 10 W/(m^2 K) y la temperatura de la zona izquierda del cuerpo plano es de 20 °C. La zona derecha del cuerpo se encuentra a 0 °C. Determinar la distribución de temperatura en las paredes y la unión, además de la potencia calorífica $\dot{Q}$ y el flujo de calor q.



CÁLCULO CON EL MÉTODO ANALÍTICO:

Datos:

$T_1 = T_{zona\ caliente} = 20 + 273.15 = 293.15$ K

$T_2 = T_{zona\ fría} = 0 + 273.15 = 273.15$ K

h = 10 W/ (m^2 K)

$k_1 = 5$ W/ (m K)

$k_2 = 0.4$ W/ (m K)

$A_1 = A_2 = 1\ m^2$

$e_1 = e_2 = 10$ cm

Calculemos primero la resistencia térmica para una configuración de capas planas paralelas:

$$R_T = \frac{1}{h_i} + \frac{e_1}{k_1} + \frac{e_2}{k_2} + \frac{1}{h_e} = \frac{1}{10} + \frac{0.1}{5} + \frac{0.1}{0.4} + \frac{1}{10} = 0.47\ \frac{m^2K}{W}$$

La conductividad global es:

$$U_T = \frac{1}{R_T} = \frac{1}{0.47\ \frac{m^2K}{W}} = 2.1276\frac{W}{m^2K}$$

La cual nos permite obtener **el flujo de calor a través de las capas:**

$$q = U_T\ (T_1 - T_2) = U_T\ (T_{zi} - T_{zd}) = \left(2.1276\frac{W}{m^2K}\right)(293.15 - 273.15)K = 42.56\frac{W}{m^2}$$

Y **la potencia calorífica** es:

$$\dot{Q} = U_T\ A\ (T_{zi} - T_{zd}) = q\ A = \left(42.56\frac{W}{m^2}\right)(1\ m^2) = 42.56\ W$$

Para las temperaturas, en las paredes interna, unión y externa será:

Pared izquierda:

$$R_T = \frac{1}{h_i}$$

$$U_T = \frac{1}{R_T} = h_i$$

$$\dot{Q} = h_i\ A\ (T_{zi} - T_i)$$

$$42.56\ W = \left(10\ \frac{W}{m^2K}\right)\ (1\ m^2)\ (293.15 - T_i)$$

$$(293.15 - T_i) = \frac{42.56\ W}{\left(10\ \frac{W}{m^2K}\right)\ (1\ m^2)} = 4.256\ K$$

$$T_i = 293.15\ K - 4.256\ K = 288.894\ K$$

En la unión:

$$R_T = \frac{e_1}{k_1}$$

$$U_T = \frac{1}{R_T} = \frac{k_1}{e_1}$$

$$\dot{Q} = \left(\frac{k_1}{e_1}\right)\ A\ (T_i - T_u)$$

$$42.56\ W = \left(\frac{5}{0.1}\right)\frac{W}{m^2K}\ (1\ m^2)\ (288.894 - T_u)$$

$$(288.894 - T_u) = \frac{42.56\ W}{\left(50\ \frac{W}{m^2K}\right)\ (1\ m^2)} = 0.8516\ K$$

$$T_u = 288.894\ K - 0.8516\ K = 288.0424\ K$$

Pared derecha:

$$R_T = \frac{e_2}{k_2}$$

$$U_T = \frac{1}{R_T} = \frac{k_2}{e_2}$$

$$\dot{Q} = \left(\frac{k_2}{e_2}\right)\ A\ (T_u - T_e)$$

$$42.56\ W = \left(\frac{0.4}{0.1}\right)\frac{W}{m^2K}\ (1\ m^2)\ (288.0424 - T_e)$$

$$(288.0424 - T_e) = \frac{42.56\ W}{\left(4\ \frac{W}{m^2K}\right)\ (1\ m^2)} = 10.64\ K$$

$$T_e = 288.0424\ K - 10.64\ K = 277.4024\ K$$

Zona derecha:

$$R_T = \frac{1}{h_e}$$

$$U_T = \frac{1}{R_T} = h_e$$

$$\dot{Q} = h_e\, A\, (T_e - T_{zd})$$

$$42.56\, W = \left(10\, \frac{W}{m^2 K}\right)\, (1\, m^2)\, (277.4024 - T_{zd})$$

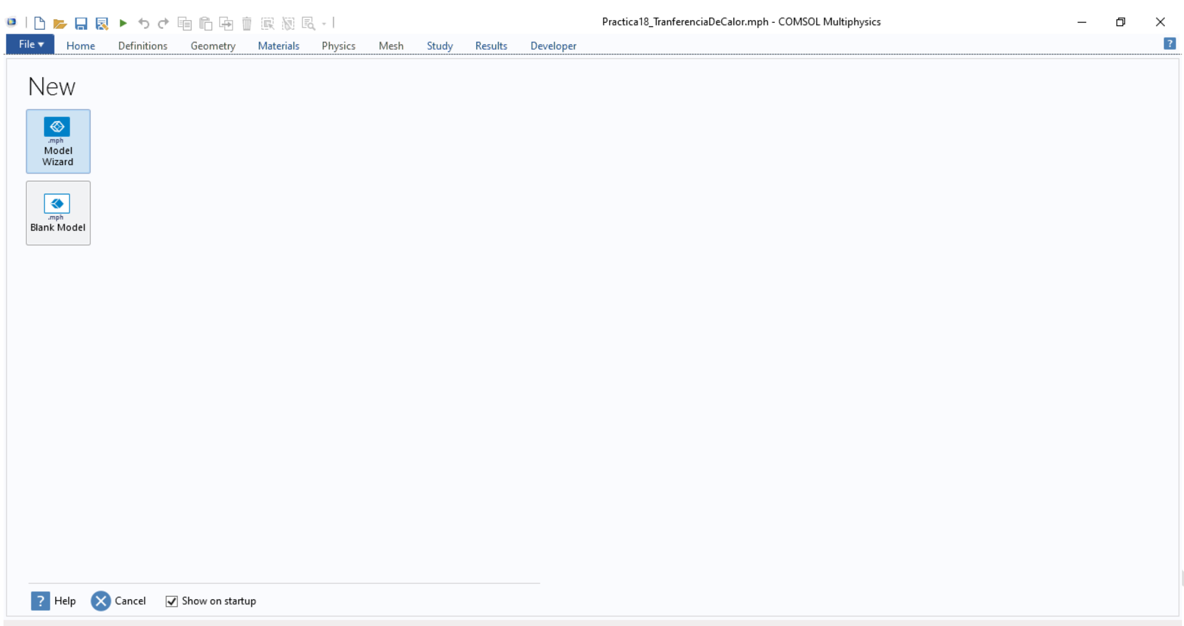
$$(277.4024 - T_{zd}) = \frac{42.56\, W}{\left(10\, \frac{W}{m^2 K}\right)\, (1\, m^2)} = 4.256\, K$$

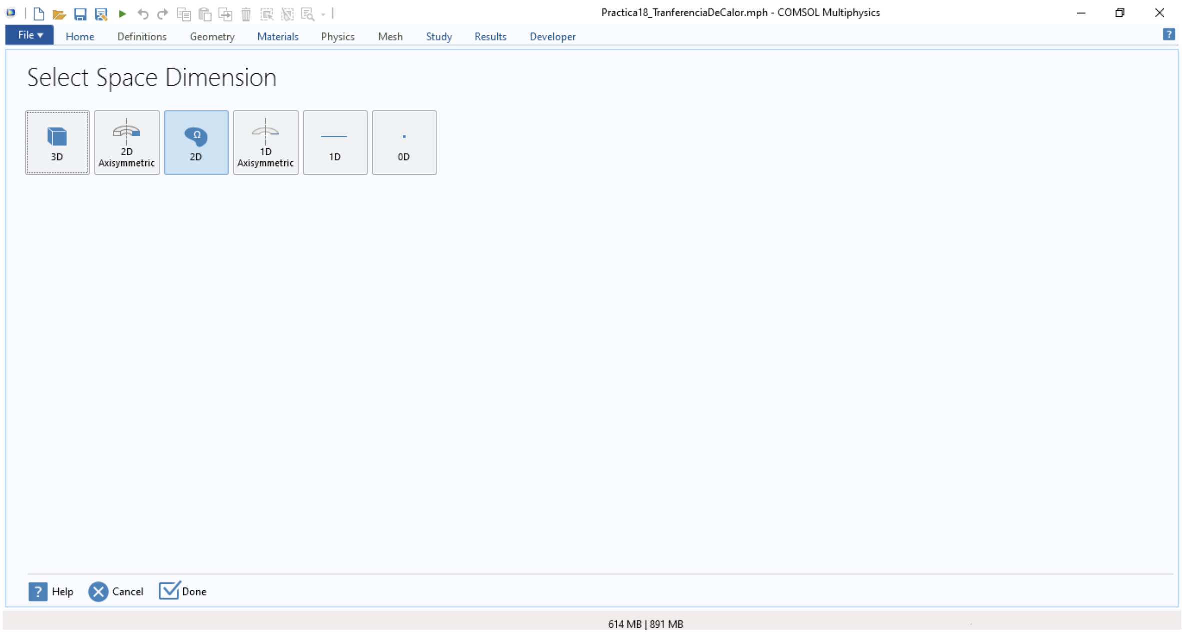
$$T_i = 277.4024\, K - 4.256\, K = 273.1464\, K$$

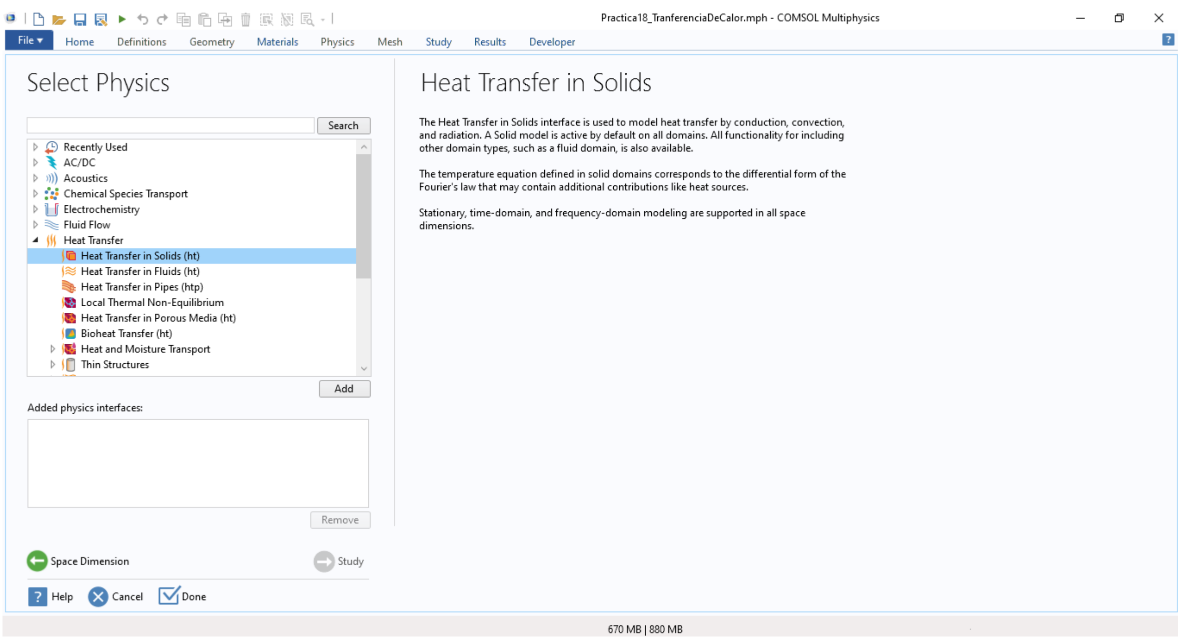
En conclusión:

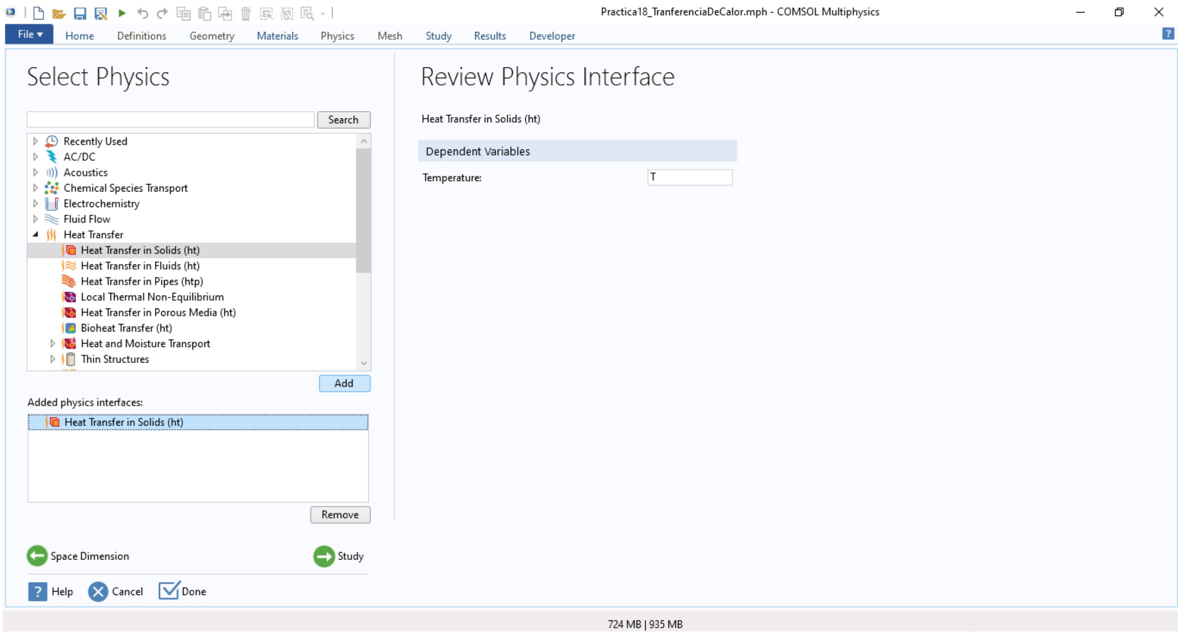
Lugar	Temperatura [K]
Zona izquierda	293.15
Pared izquierda	288.894
Unión	288.0424
Pared derecha	277.4024
Zona derecha	273.1464

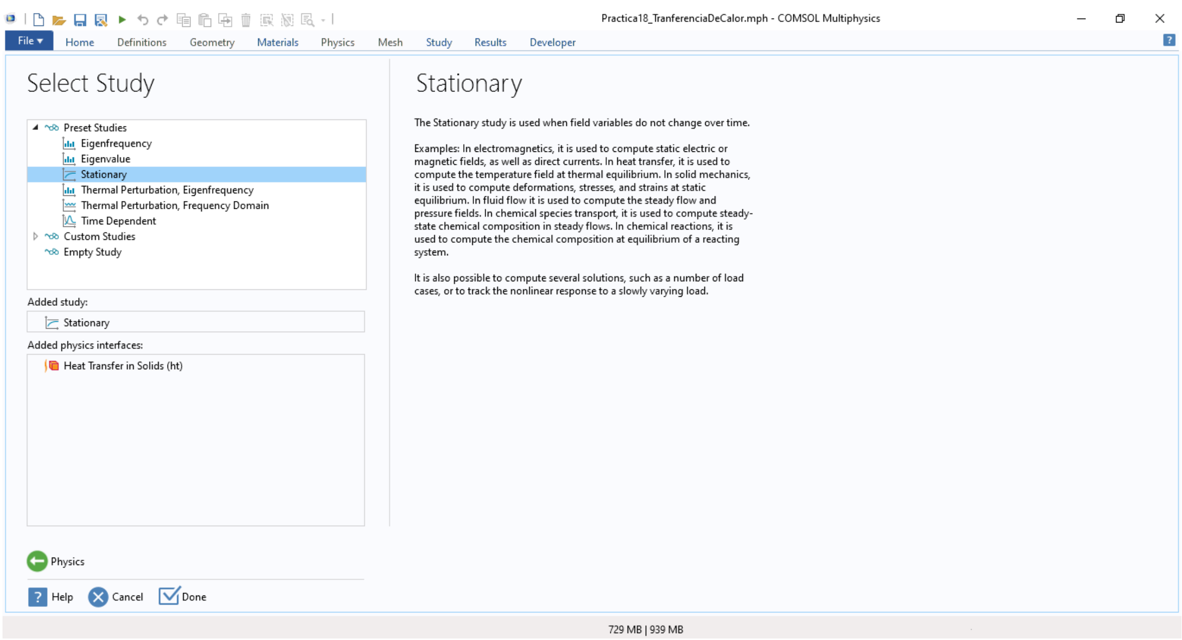
CREACIÓN DE LA PIEZA EN COMSOL:

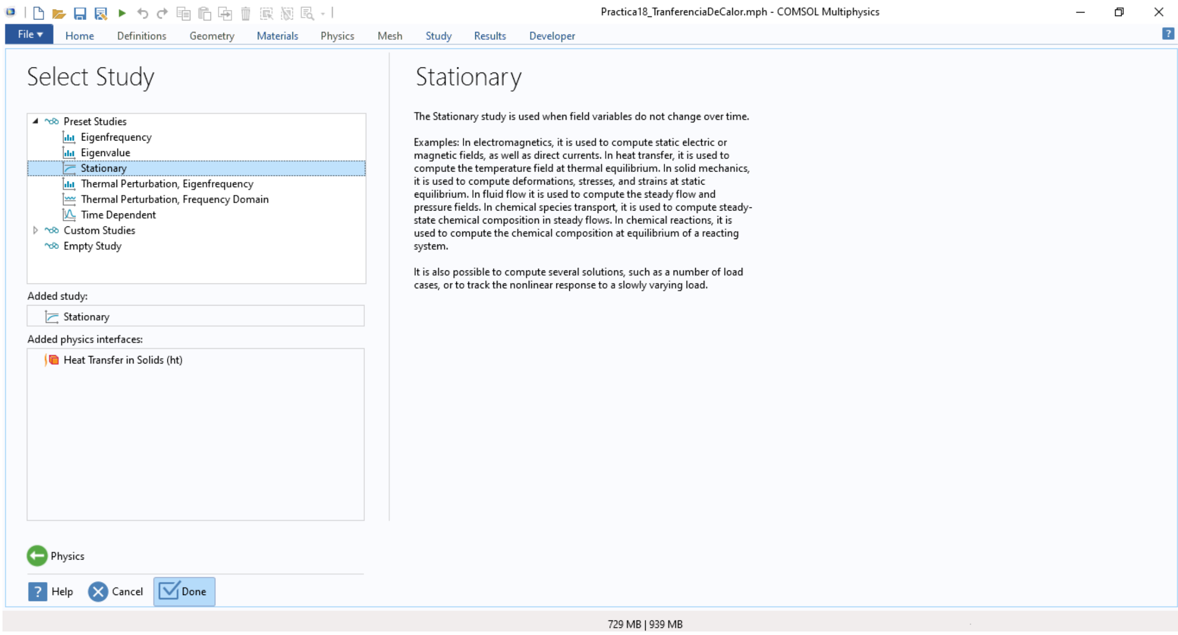


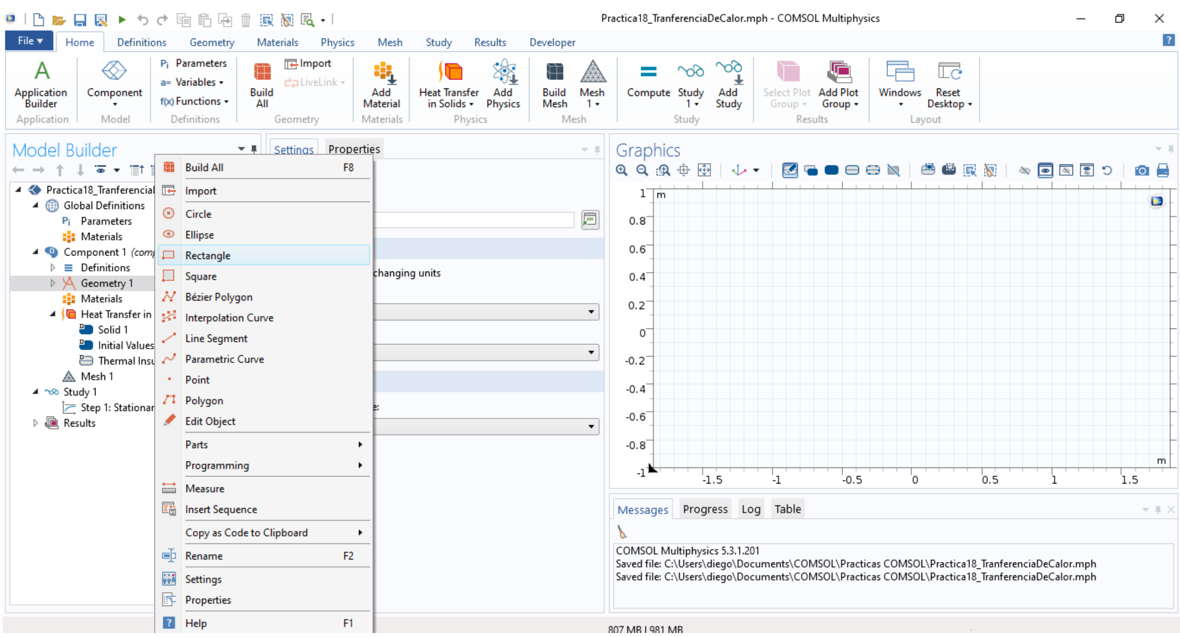

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Space Dimension
3D
2D Axisymmetric
2D
1D Axisymmetric
1D
0D
Help
Cancel
Done
614 MB | 891 MB

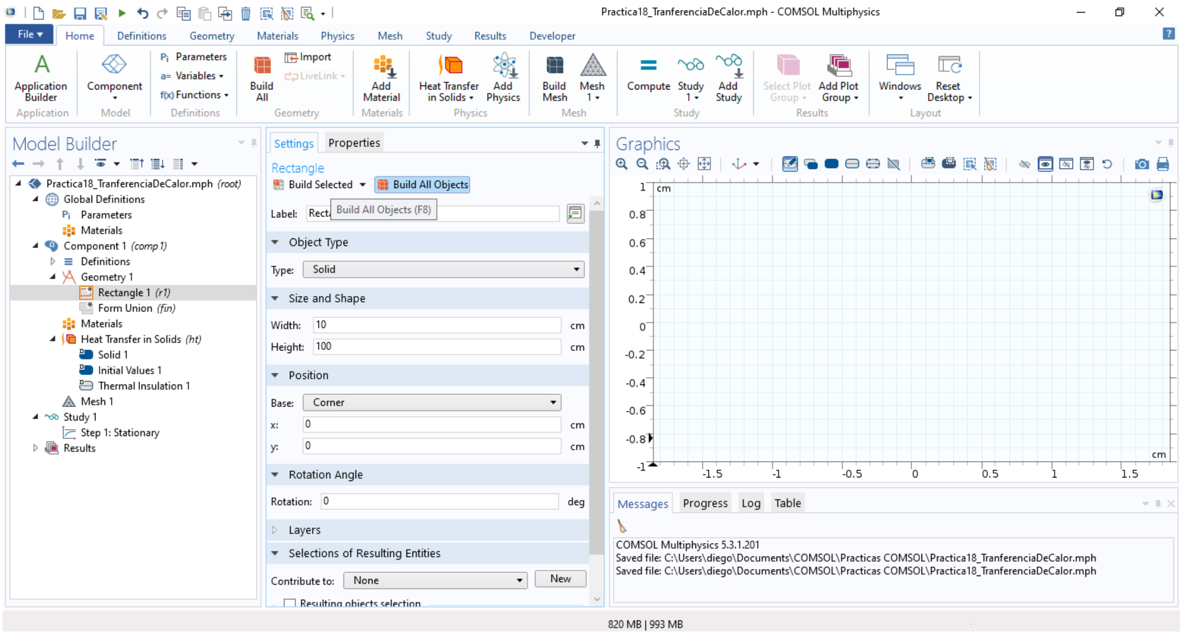

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Physics
Search
Recently Used
AC/DC
Acoustics
Chemical Species Transport
Electrochemistry
Fluid Flow
Heat Transfer
Heat Transfer in Solids (ht)
Heat Transfer in Fluids (ht)
Heat Transfer in Pipes (htp)
Local Thermal Non-Equilibrium
Heat Transfer in Porous Media (ht)
Bioheat Transfer (ht)
Heat and Moisture Transport
Thin Structures
Add
Added physics interfaces:
Remove
Space Dimension
Study
Help
Cancel
Done
Heat Transfer in Solids
The Heat Transfer in Solids interface is used to model heat transfer by conduction, convection, and radiation. A Solid model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available.
The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.
Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.
670 MB | 880 MB


Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Physics
Search
Recently Used
AC/DC
Acoustics
Chemical Species Transport
Electrochemistry
Fluid Flow
Heat Transfer
Heat Transfer in Solids (ht)
Heat Transfer in Fluids (ht)
Heat Transfer in Pipes (htp)
Local Thermal Non-Equilibrium
Heat Transfer in Porous Media (ht)
Bioheat Transfer (ht)
Heat and Moisture Transport
Thin Structures
Add
Added physics interfaces:
Heat Transfer in Solids (ht)
Remove
Space Dimension
Study
Help
Cancel
Done
Review Physics Interface
Heat Transfer in Solids (ht)
Dependent Variables
Temperature:
T
724 MB | 935 MB

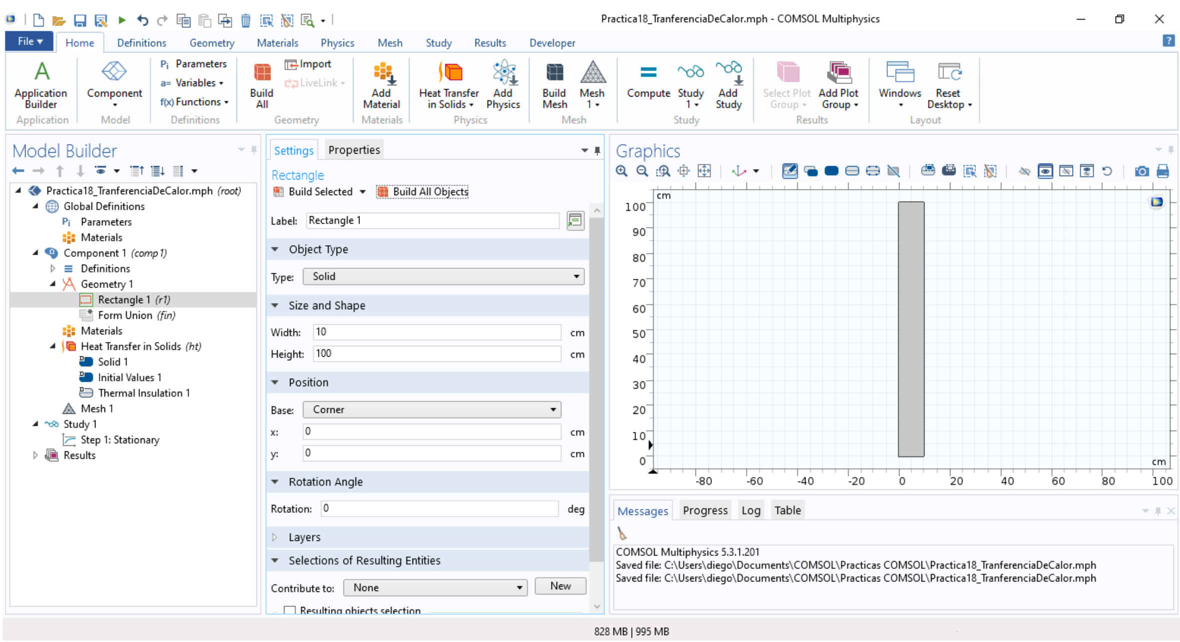

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Study
Preset Studies
Eigenfrequency
Eigenvalue
Stationary
Thermal Perturbation, Eigenfrequency
Thermal Perturbation, Frequency Domain
Time Dependent
Custom Studies
Empty Study
Added study:
Stationary
Added physics interfaces:
Heat Transfer in Solids (ht)
Physics
Help
Cancel
Done
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.
729 MB | 939 MB

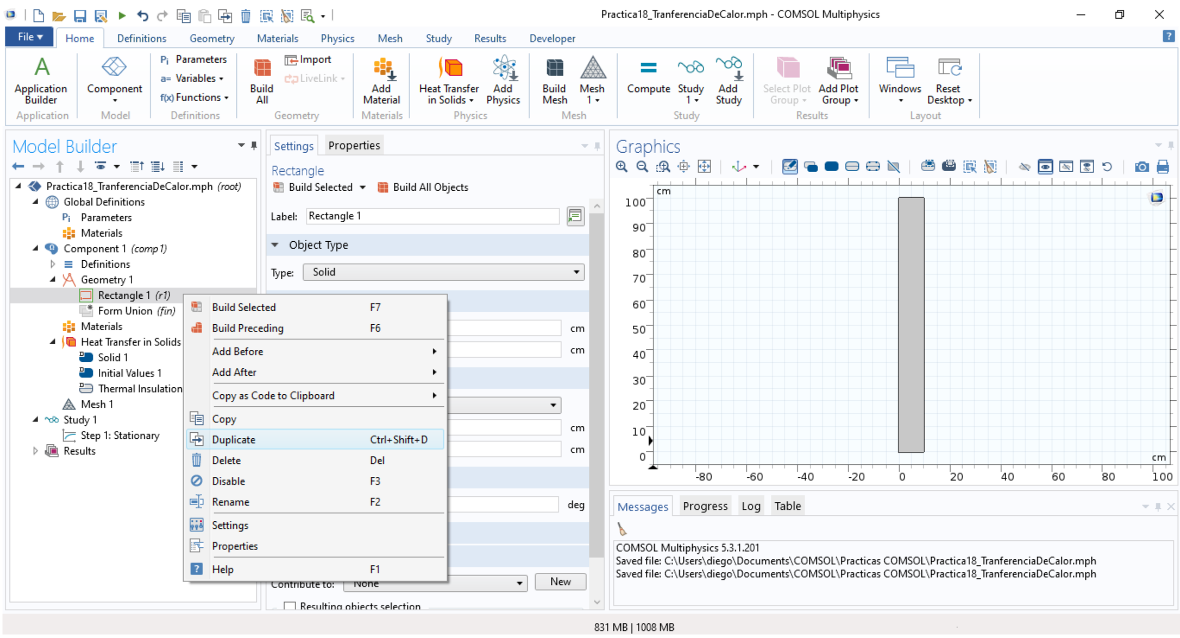

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Select Study
Preset Studies
Eigenfrequency
Eigenvalue
Stationary
Thermal Perturbation, Eigenfrequency
Thermal Perturbation, Frequency Domain
Time Dependent
Custom Studies
Empty Study
Added study:
Stationary
Added physics interfaces:
Heat Transfer in Solids (ht)
Physics
Help
Cancel
Done
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.
729 MB | 939 MB

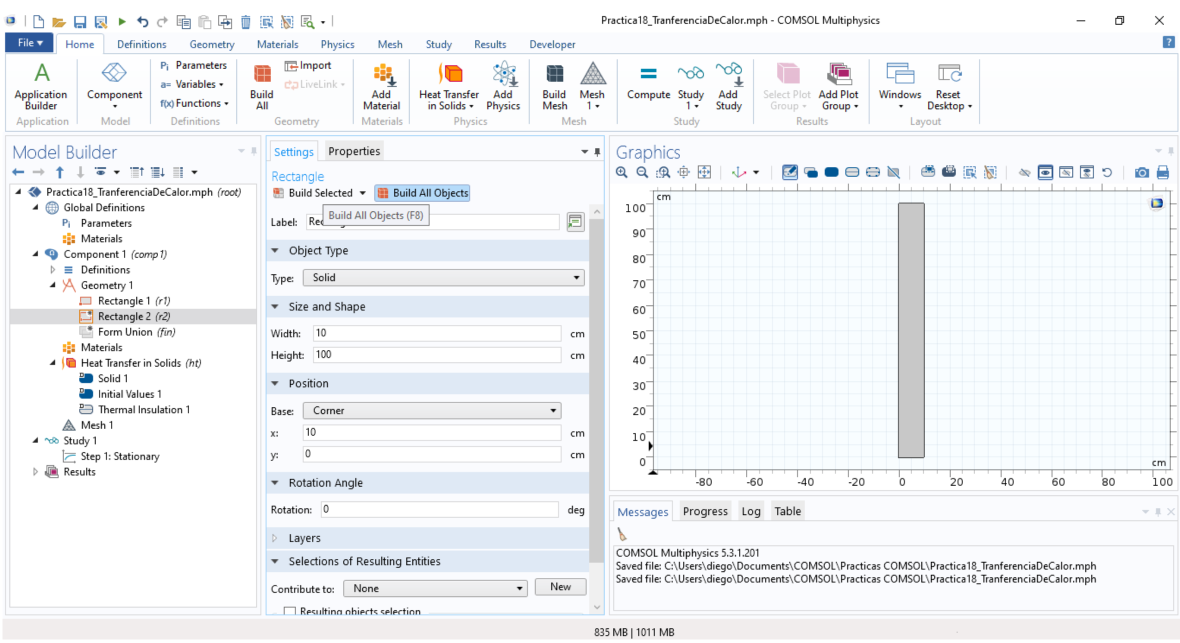

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File
Home
Definitions
Geometry
Materials
Physics
Mesh
Study
Results
Developer
Application Builder
Component
Parameters
Variables
Functions
Build All
Import
LiveLink
Add Material
Heat Transfer in Solids
Add Physics
Build Mesh
Mesh 1
Compute
Study 1
Add Study
Select Plot Group
Add Plot Group
Windows
Reset Desktop
Model Builder
Practica18_Tranferencia
Global Definitions
Parameters
Materials
Component 1
Definitions
Geometry 1
Materials
Heat Transfer in
Solid 1
Initial Values
Thermal Insu
Mesh 1
Study 1
Step 1: Stationar
Results
Build All F8
Import
Circle
Ellipse
Rectangle
Square
Bézier Polygon
Interpolation Curve
Line Segment
Parametric Curve
Point
Polygon
Edit Object
Parts
Programming
Measure
Insert Sequence
Copy as Code to Clipboard
Rename F2
Settings
Properties
Help F1
Settings
Properties
changing units
Graphics
Messages
Progress
Log
Table
COMSOL Multiphysics 5.3.1.201
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica18_TranferenciaDeCalor.mph
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica18_TranferenciaDeCalor.mph
807 MB | 981 MB



La placa tiene un espesor de 10 cm, una altura de 1 metro, osea 100 cm y una base de 1 metro igual, por lo que tendrá un área de 1 m^2.

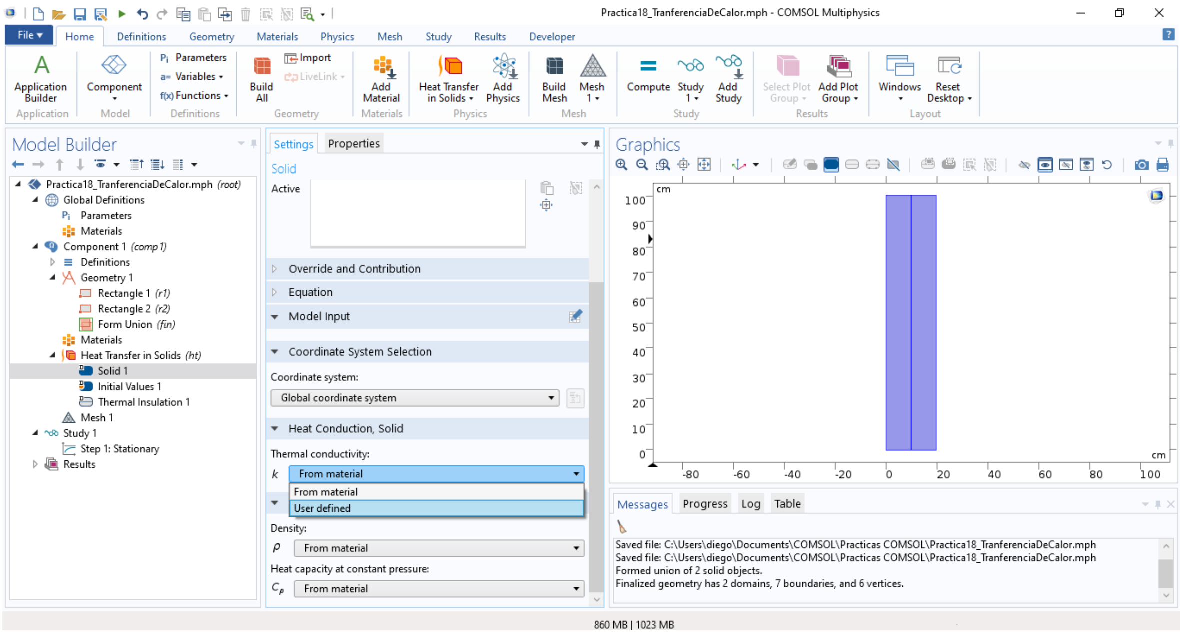




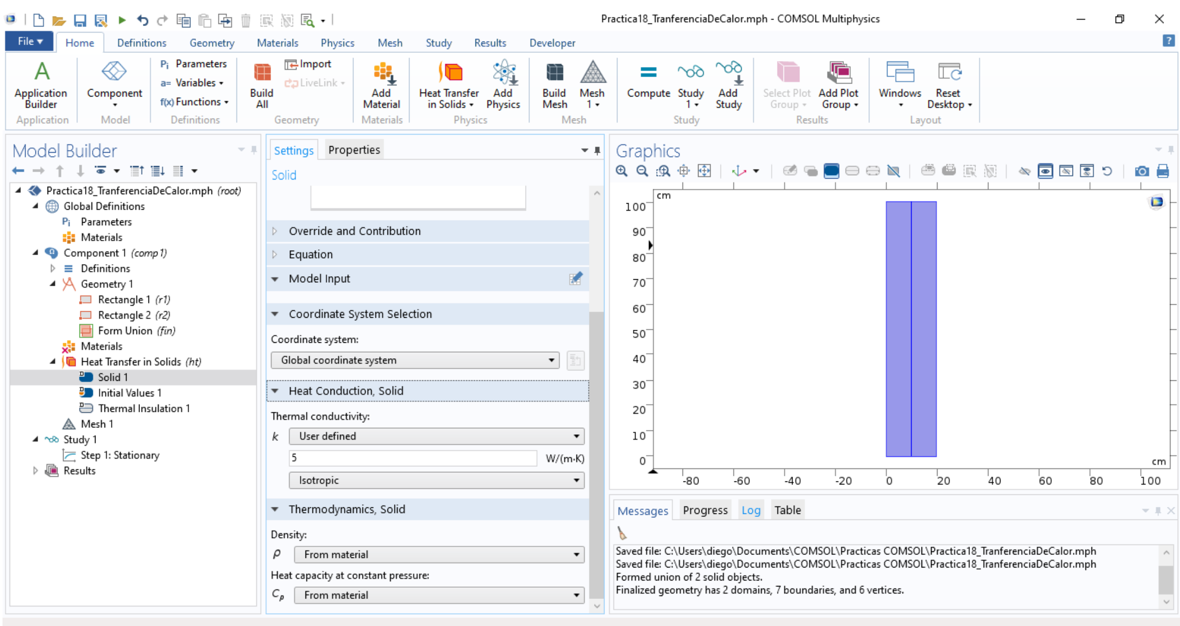


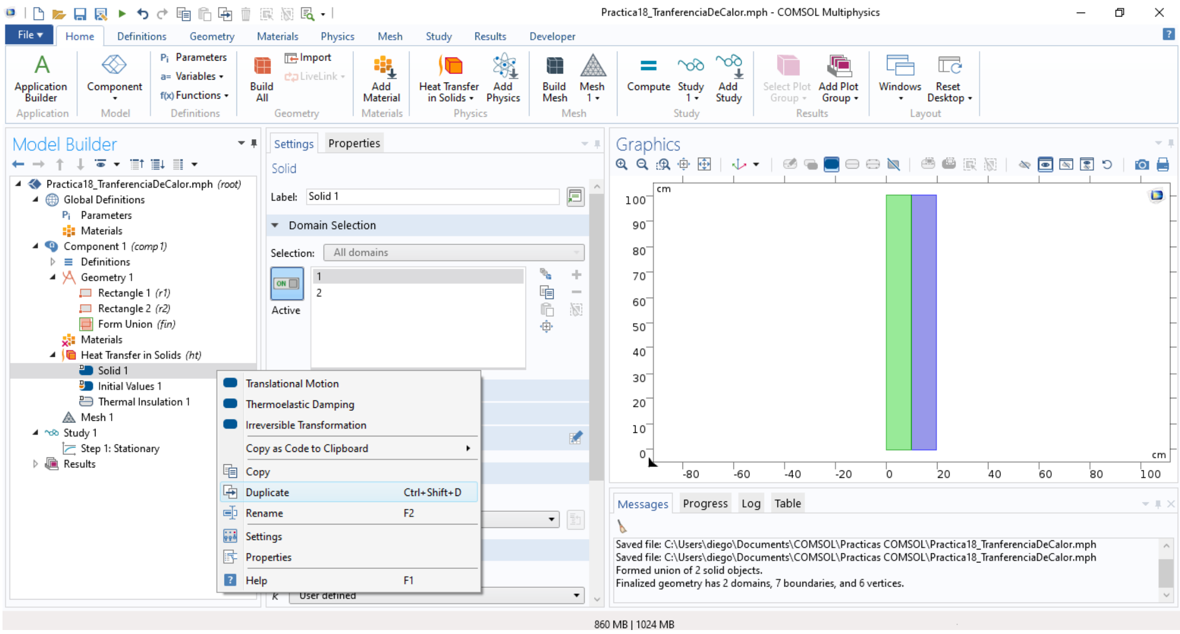
Y pongo otra placa de las mismas dimensiones, pero va a tener material diferente cada una.

ANÁLISIS DE TEMPERATURA EN COMSOL:

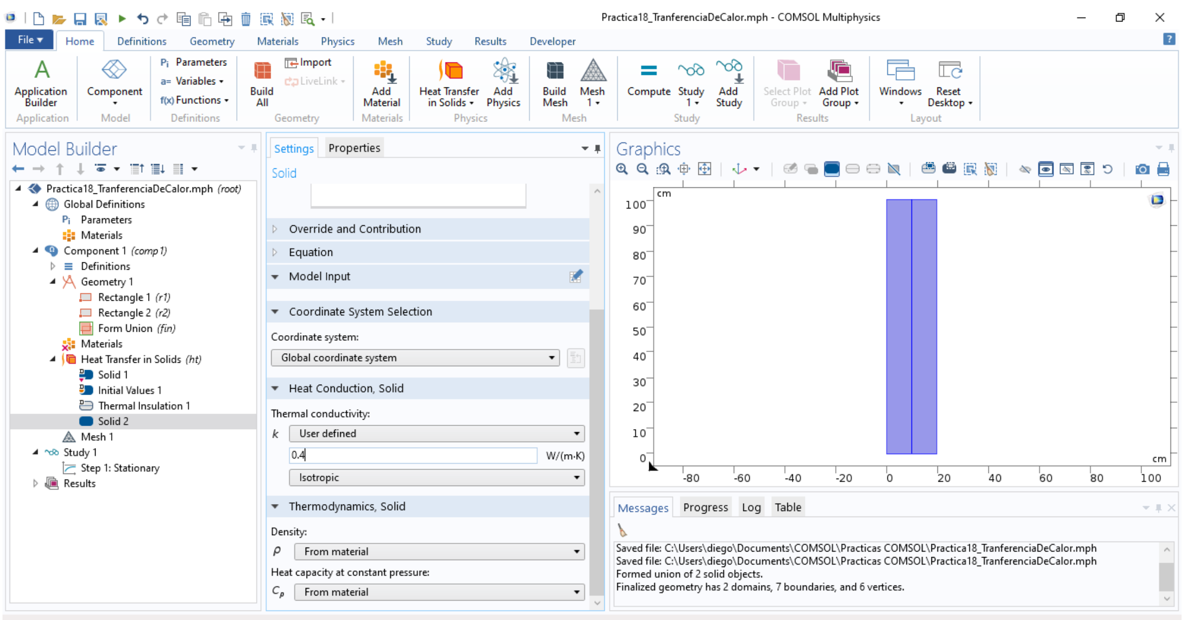


En la primera placa la conductividad es de k =5.

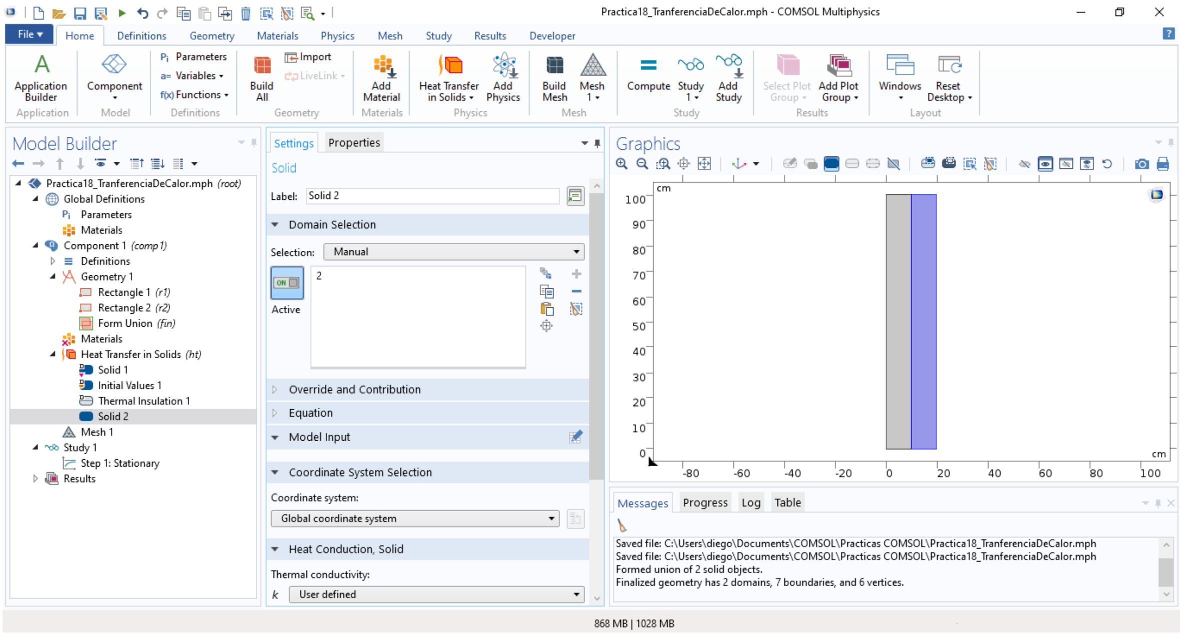


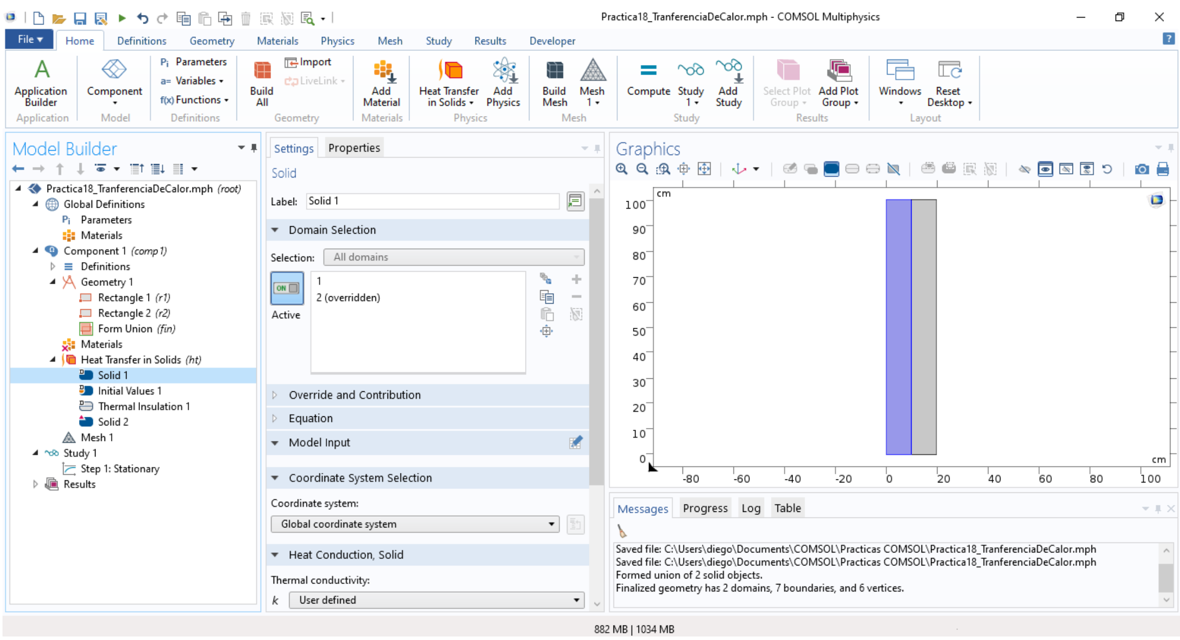


El solid 1 está por default, pero debo hacer otro para sobre escribir la k de la placa 2 que es de k=0.4.

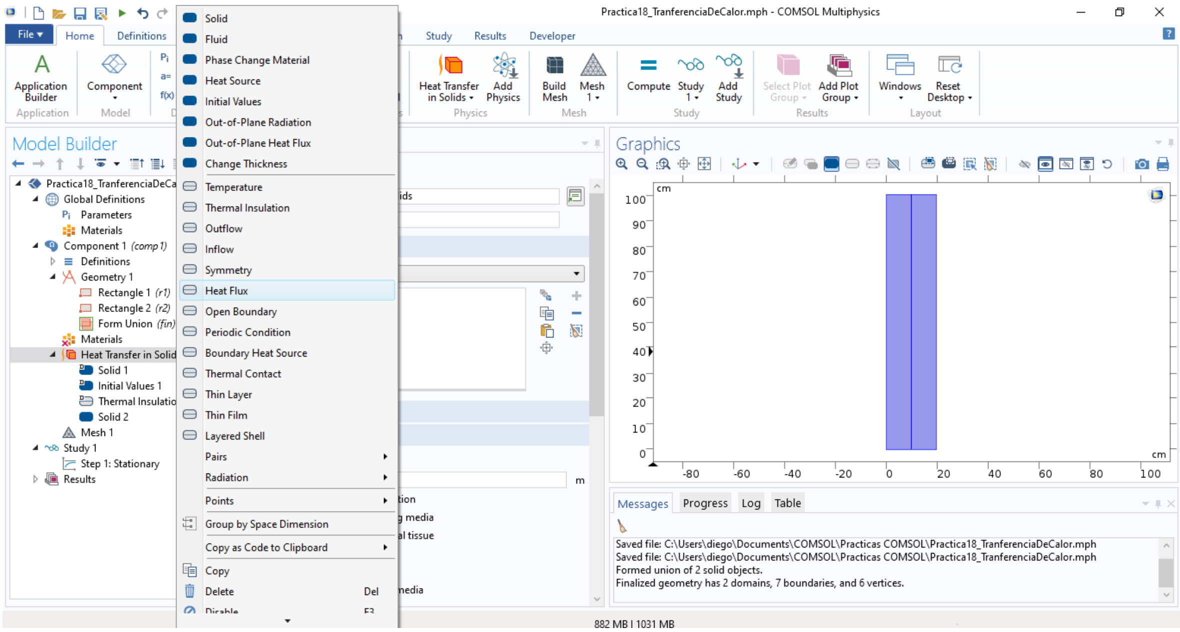


Y deseleccionamos el numero 1 para que esto solo se lo asigne a la placa 2.

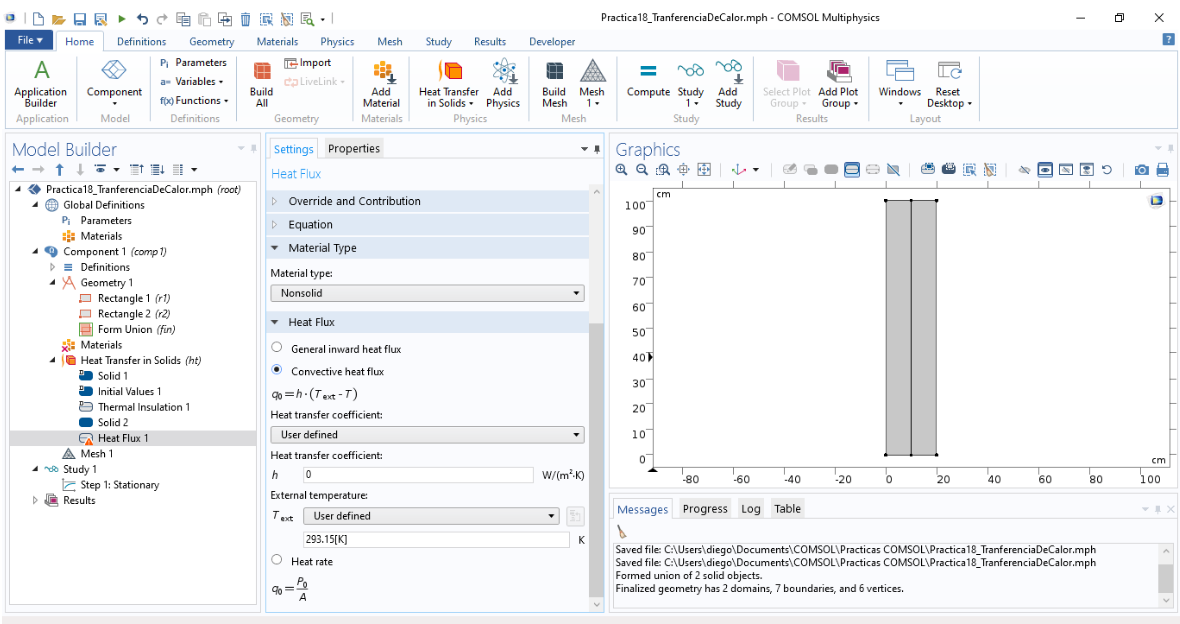




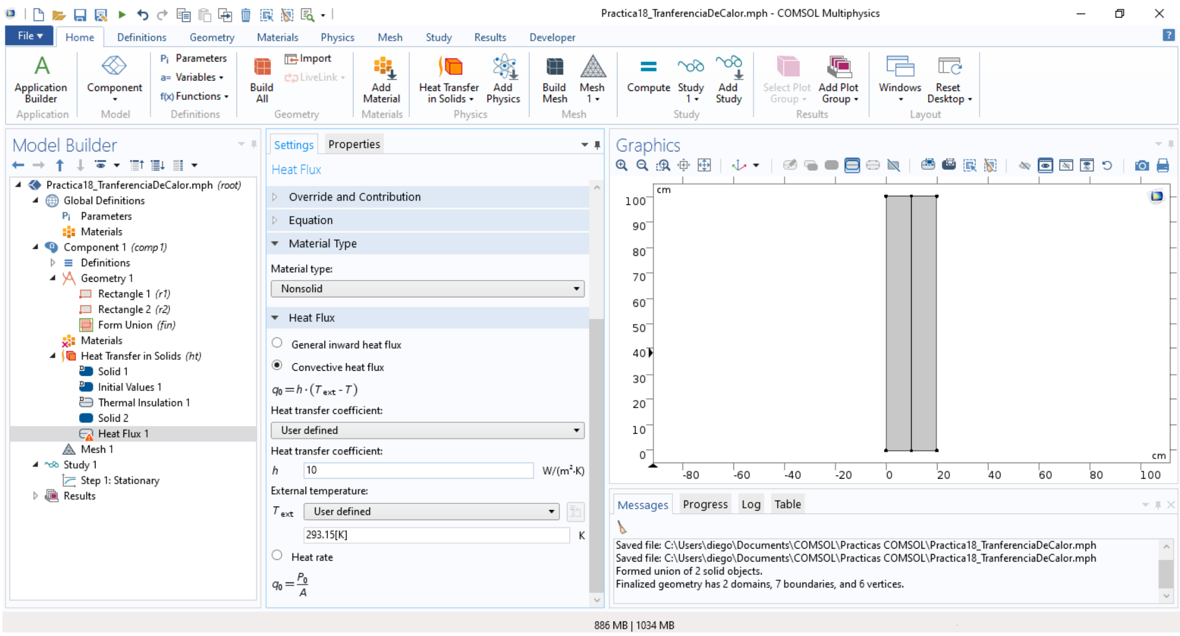
Ya que haga esto sale overridden en el 2 porque ese tiene la propiedad del solid 2.



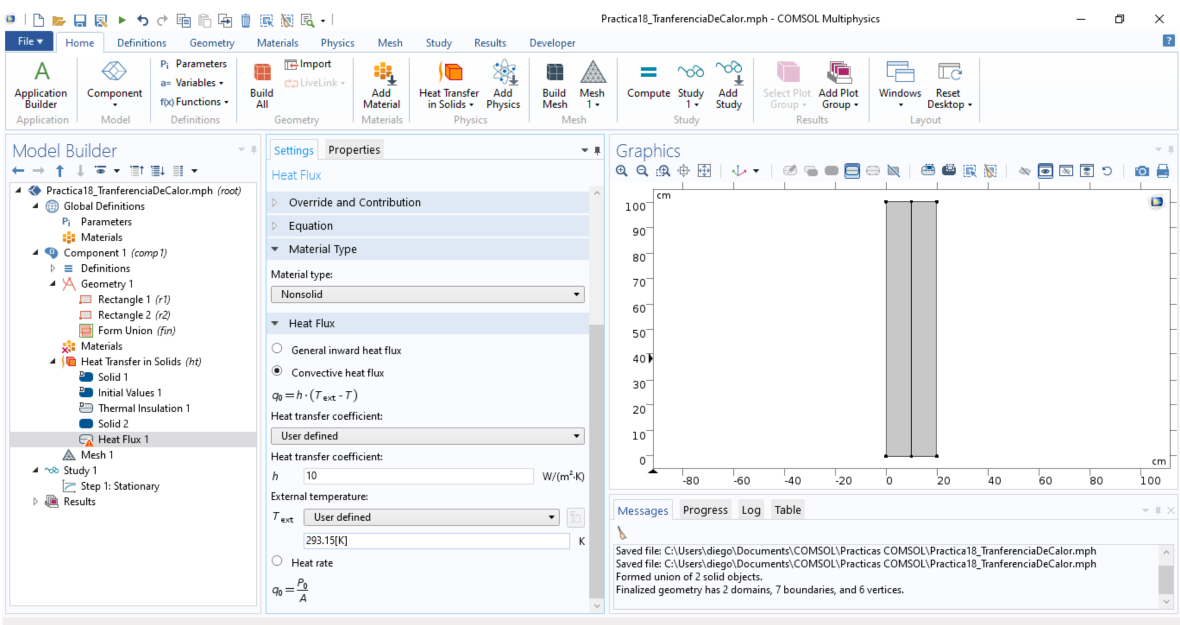
Luego elegimos que el flujo es de tipo convectivo.



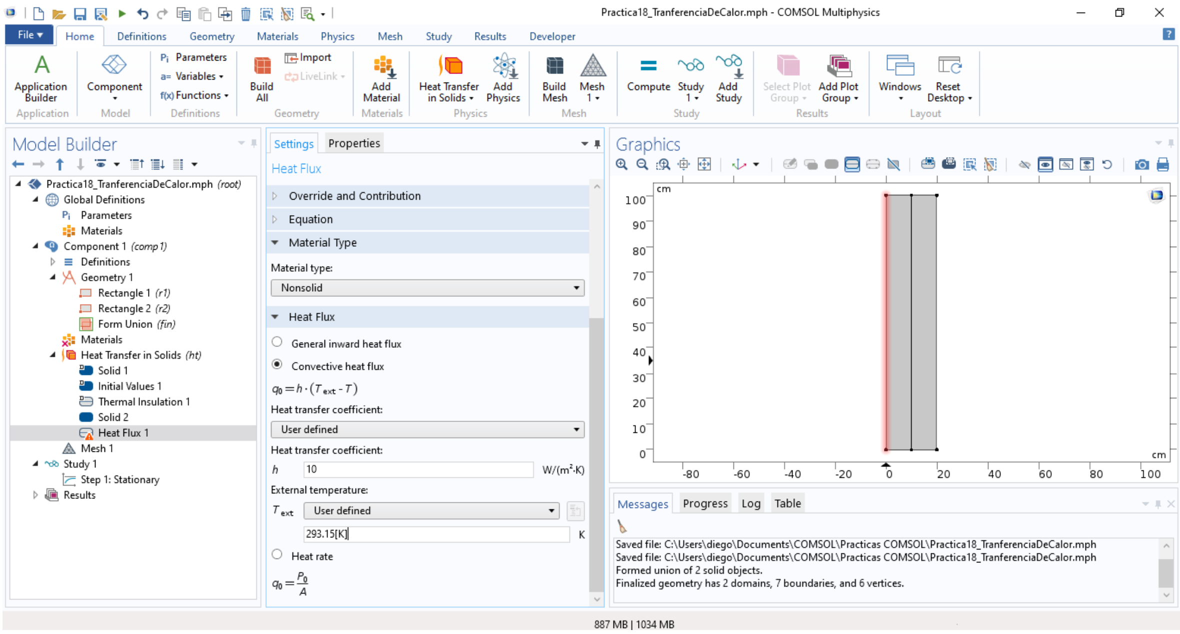
Esto es para indicar el aire alrededor de las placas.



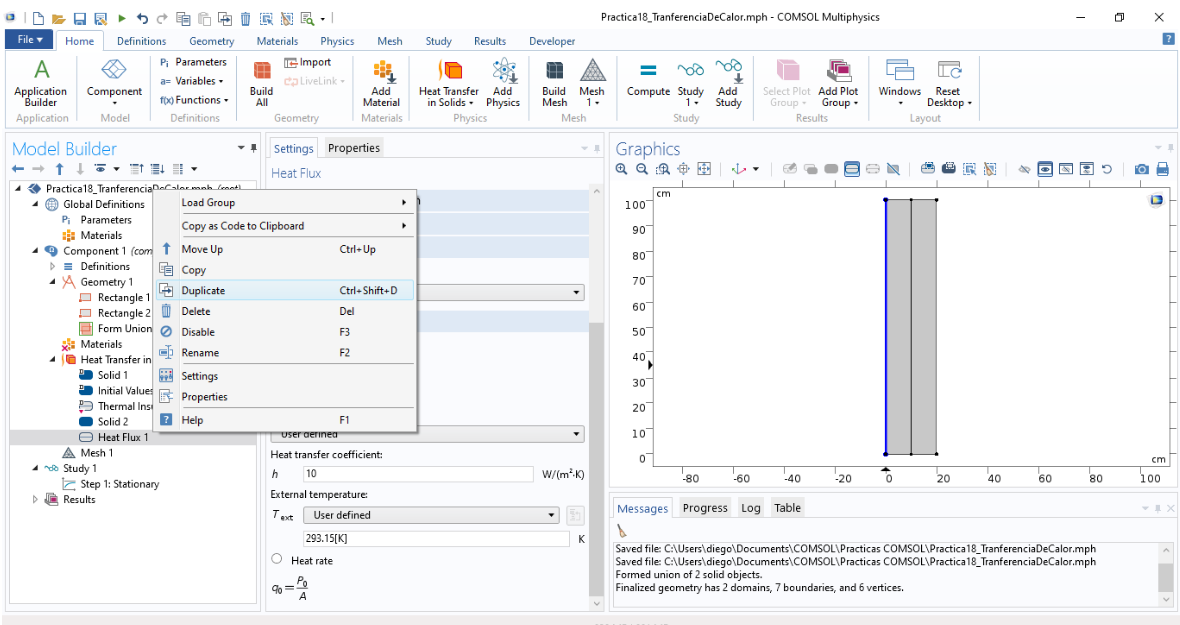
Y después ponemos la temperatura en Kelvin.

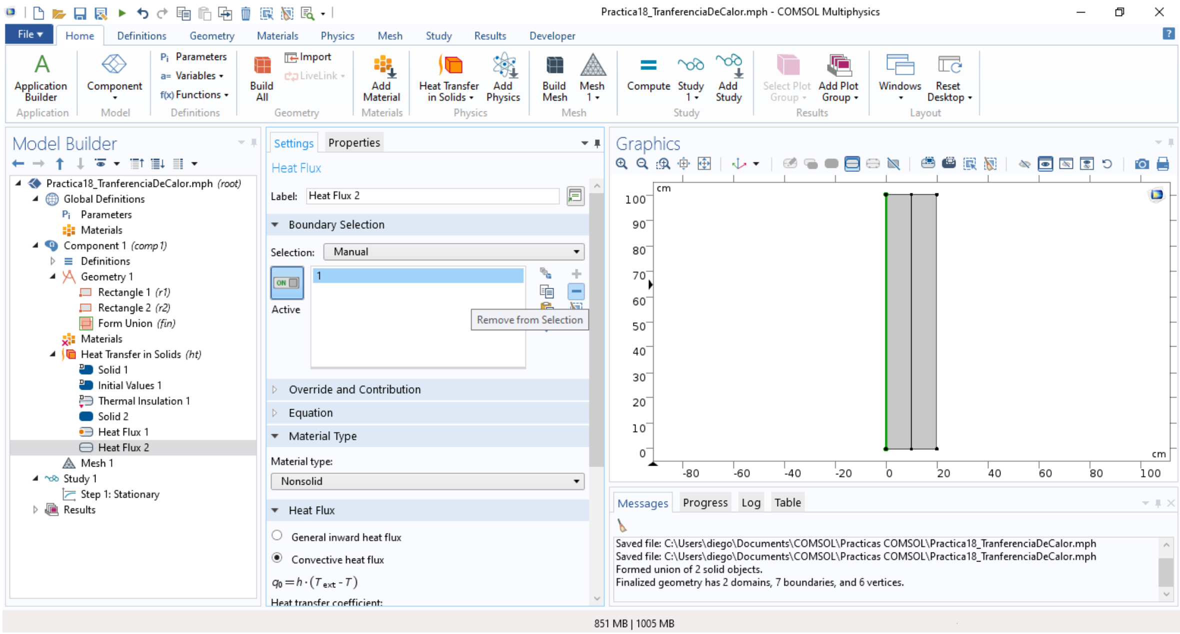


Luego indicamos donde está en contacto el flujo convectivo, osea el aire.

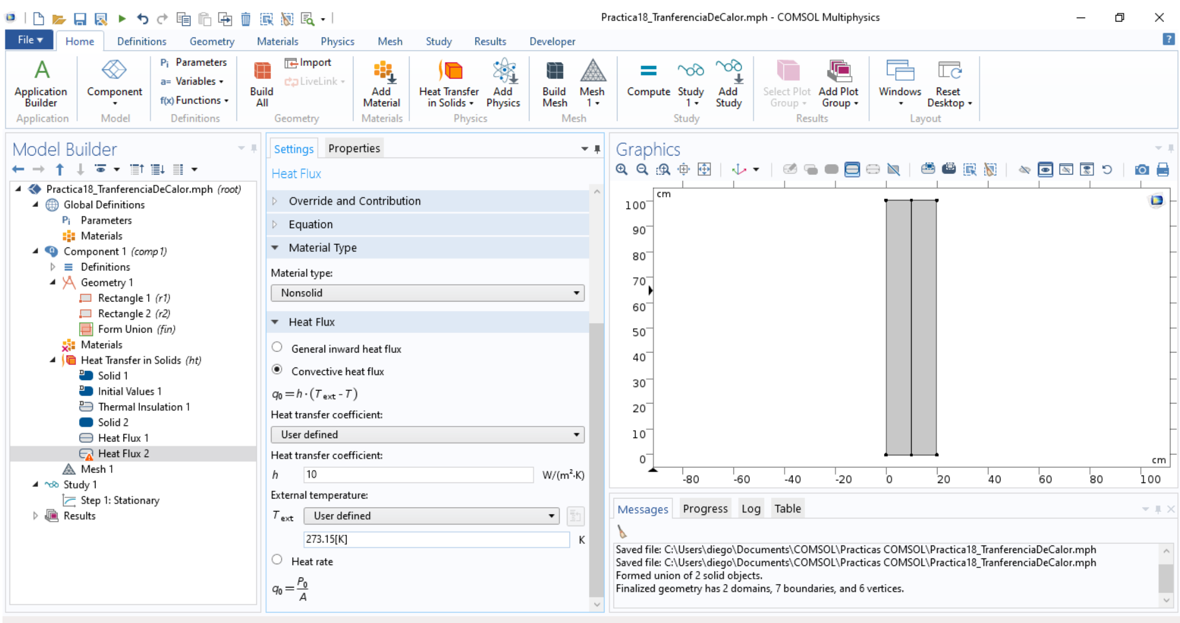


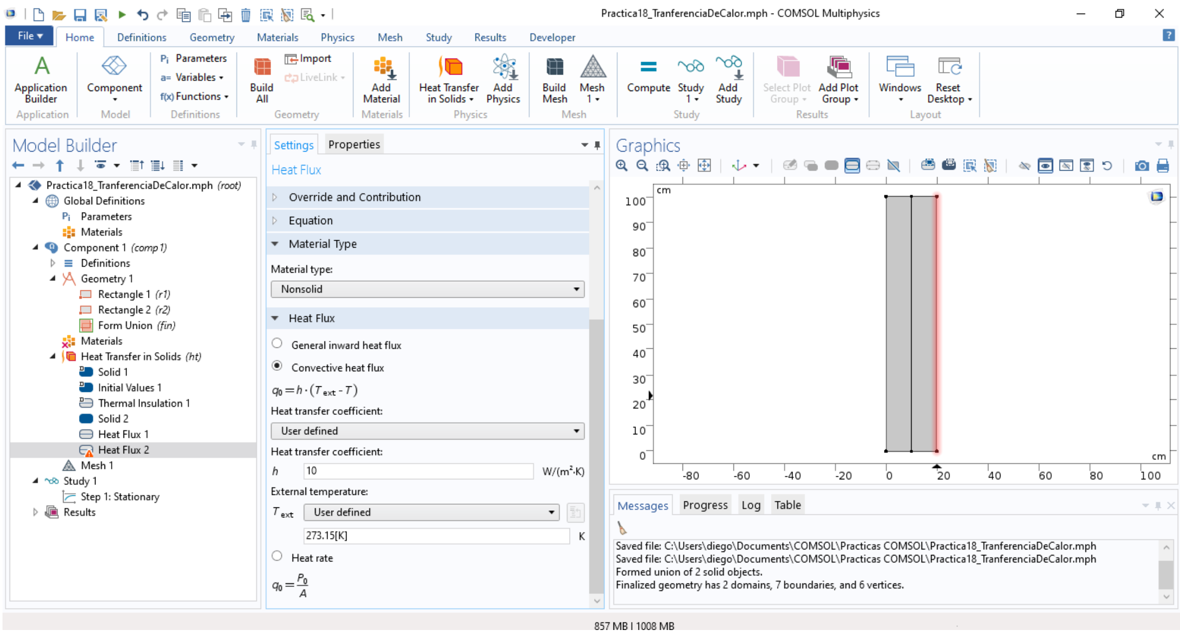
Debo ahora poner otro heat flux, pero en la otra pared de las placas.

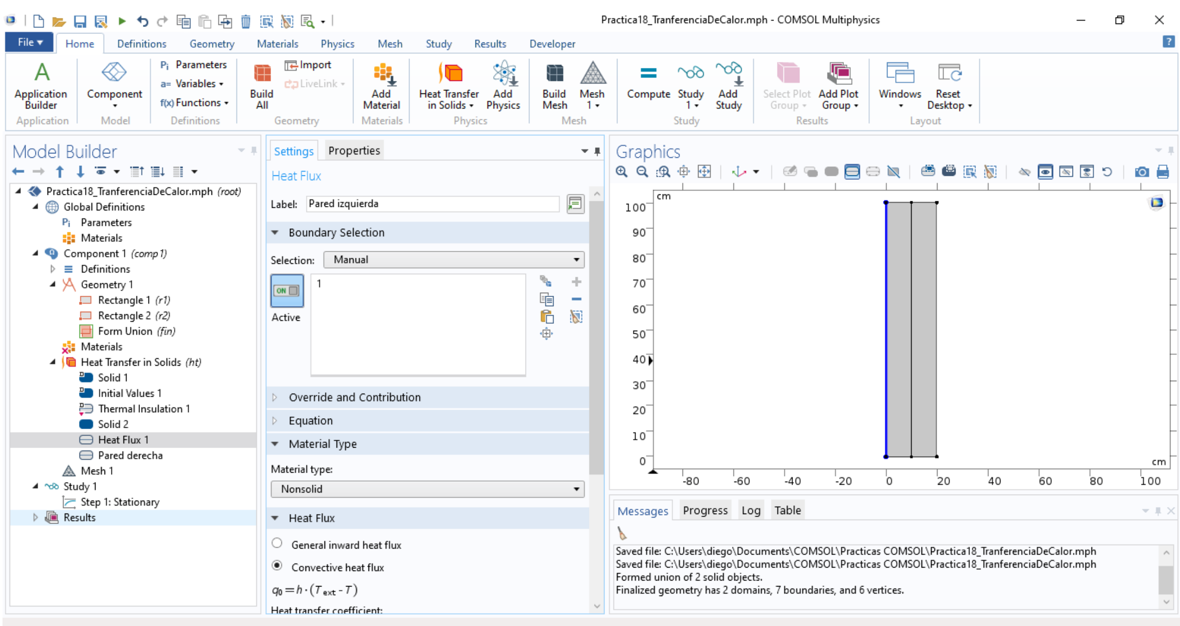




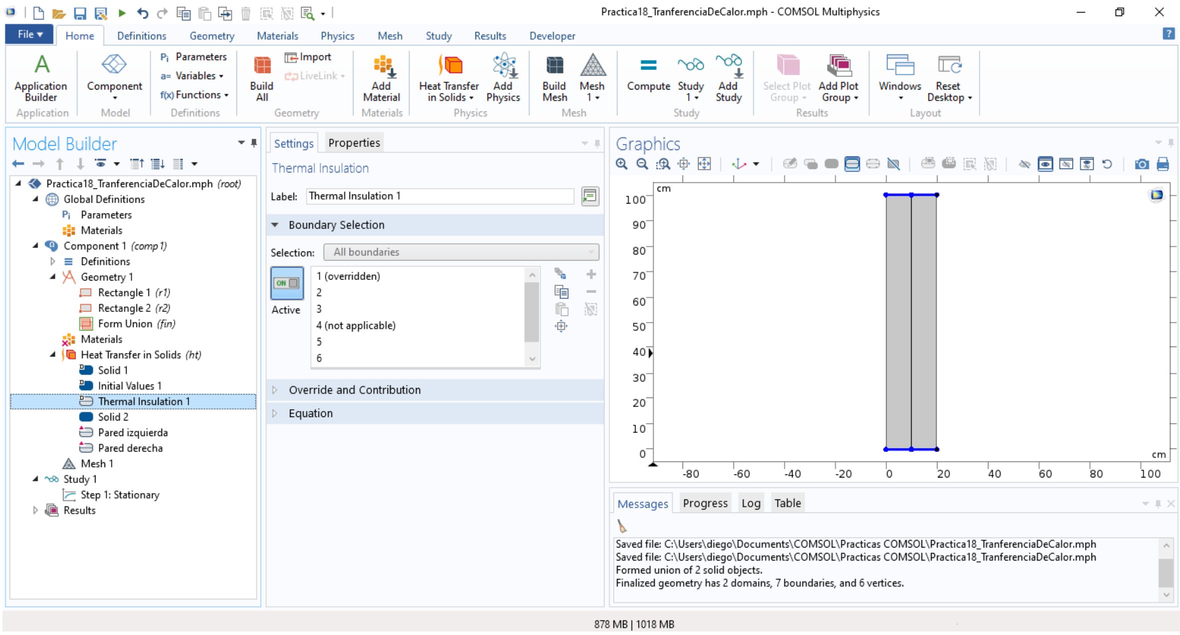
Lo que cambia en este lado es la temperatura.





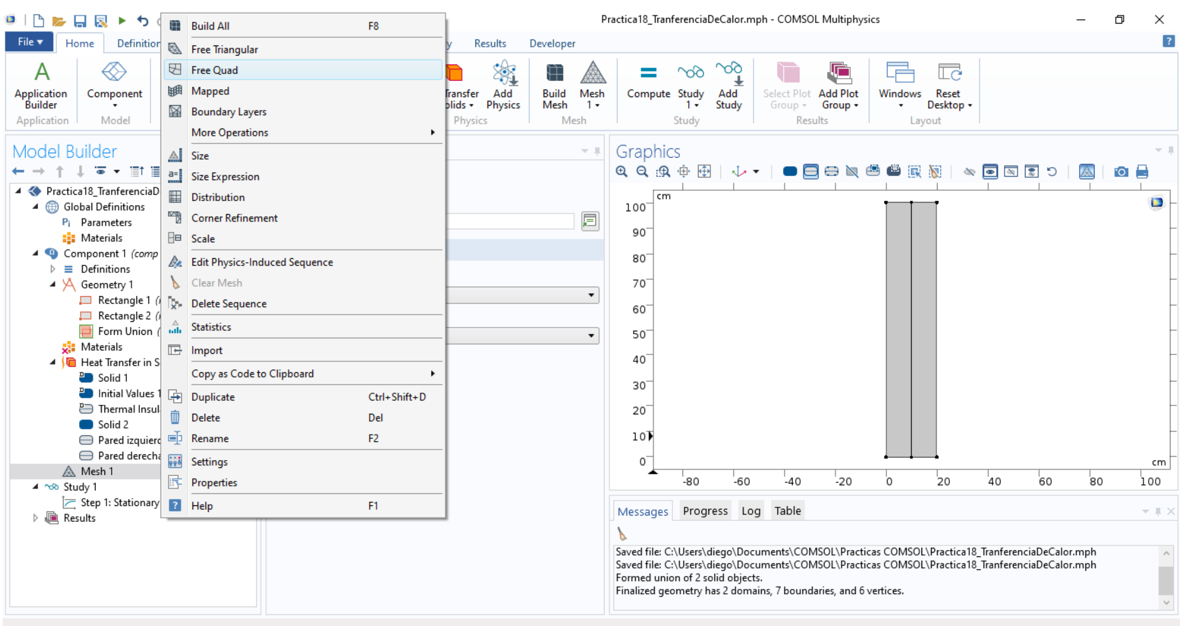


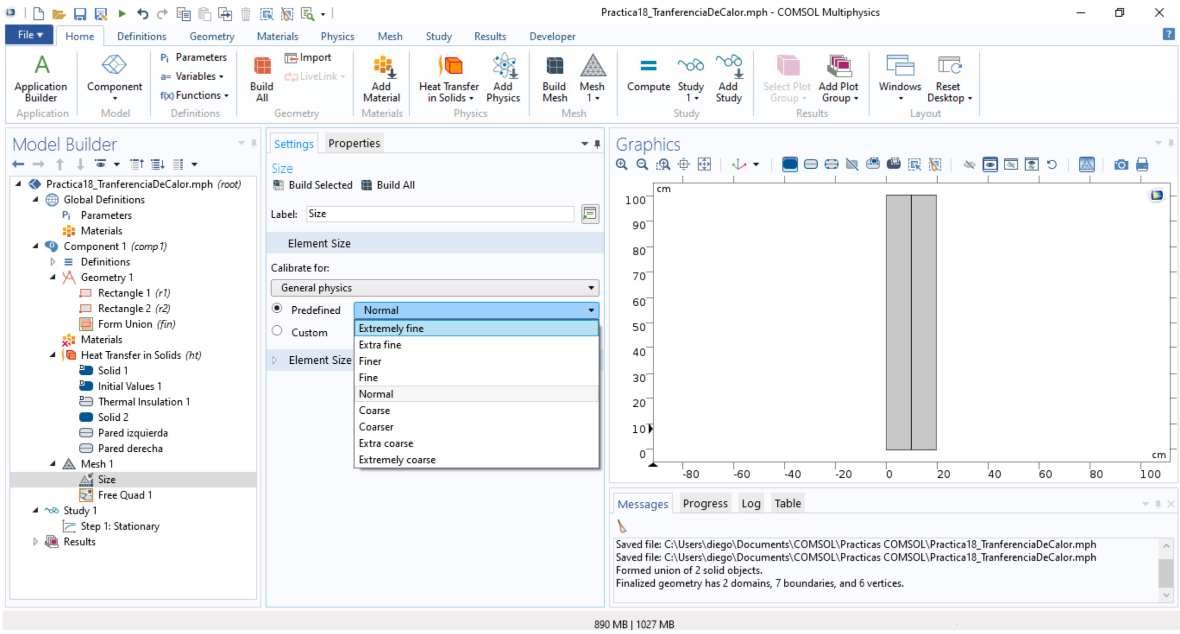
Esto coloca por default asilamiento en la pared de arriba y abajo porque en esa parte no ocurrirá transferencia de calor, esto lo hace por default el programa cuando en esas paredes no lo indiqué ningún heat flux.



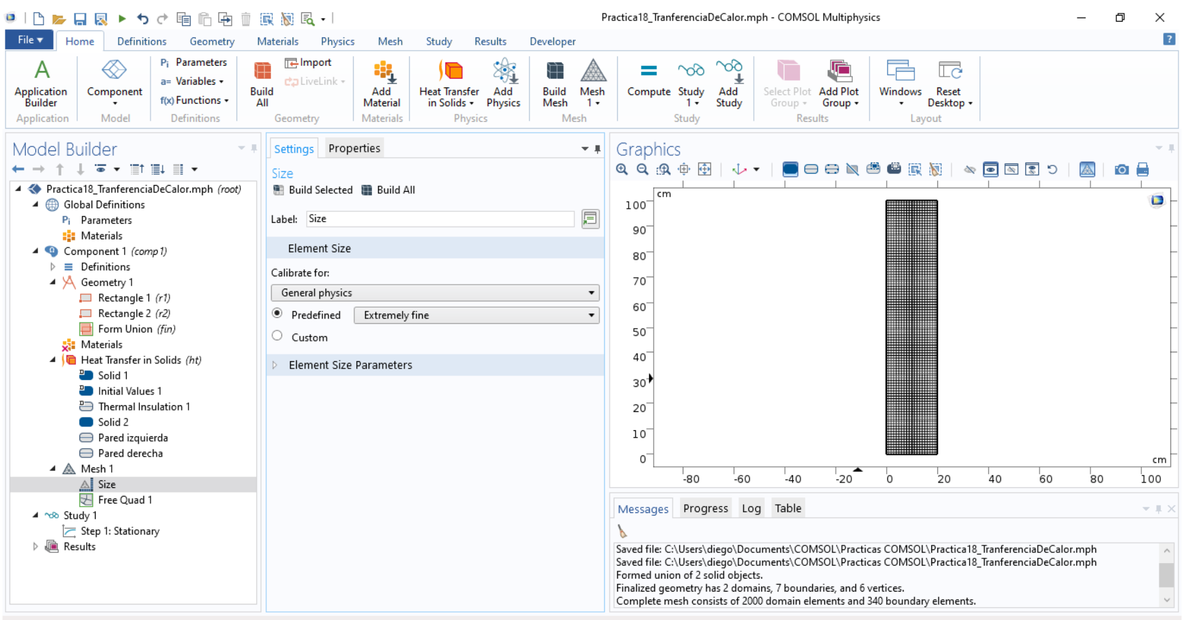
RESULTADO DEL ELEMENTO FINITO EN COMSOL:

En este caso elijo Free Quadr porque como es un elemento rectangular este tipo de malla lo maneja mejor.

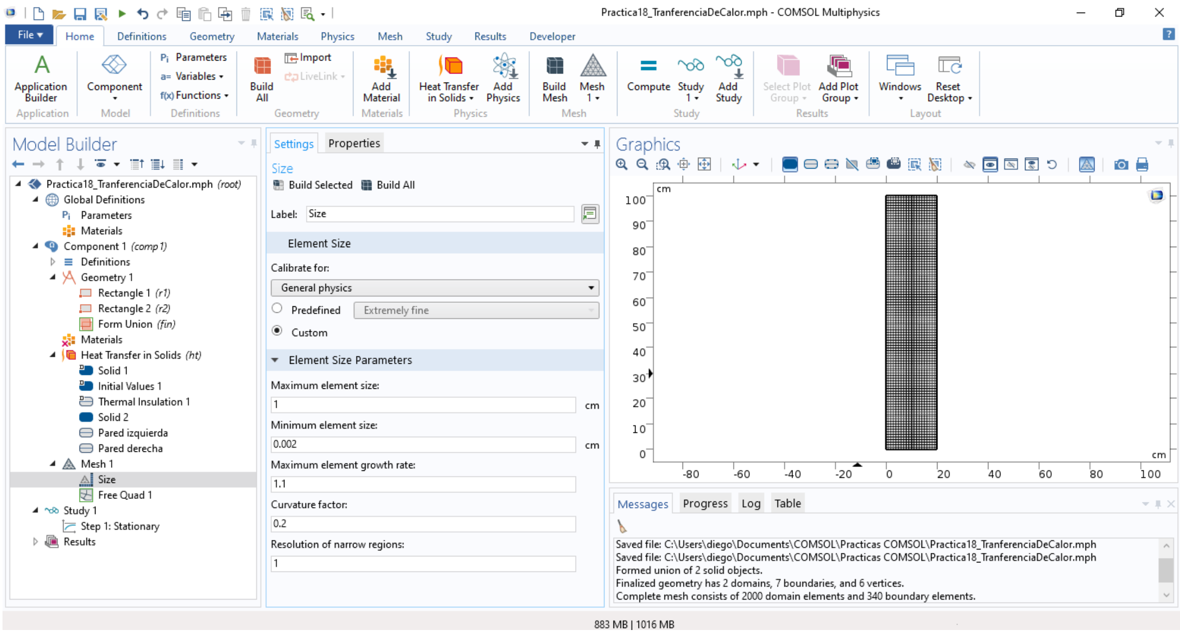


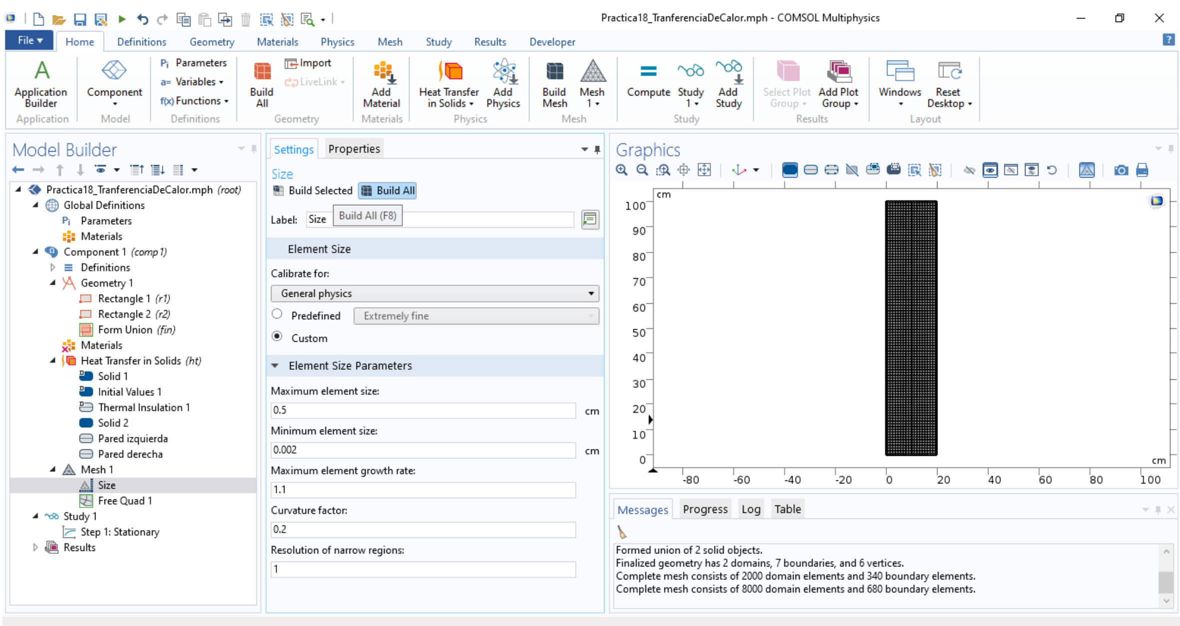


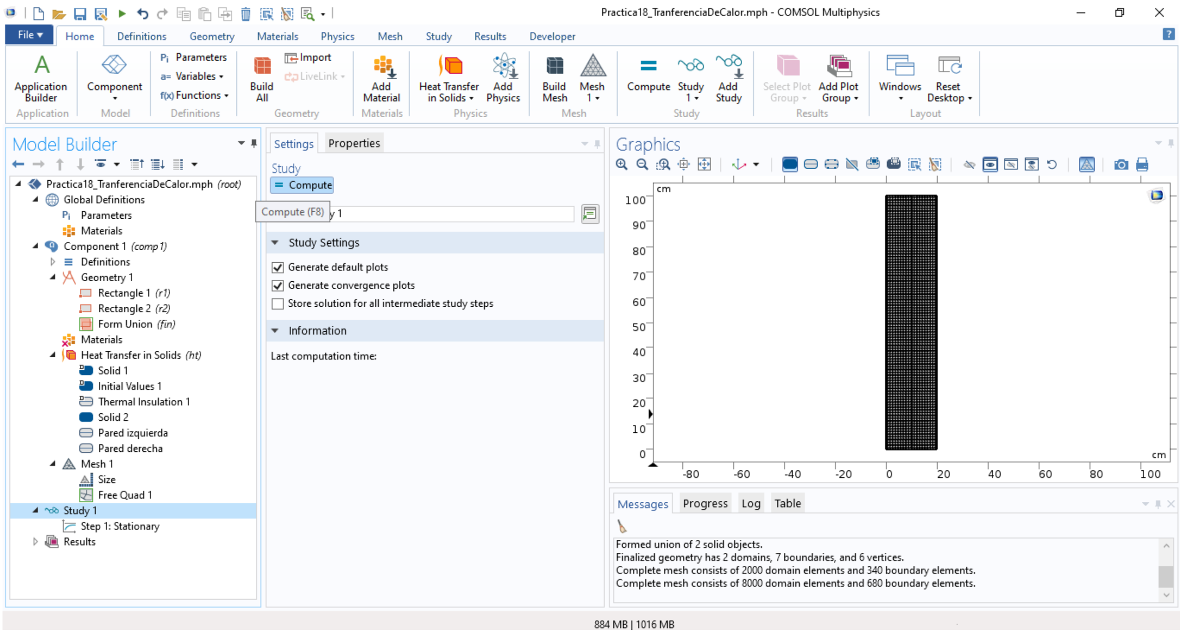
No importa que malla le demos porque este tipo de problemas no va a tardar mucho en resolverlos.

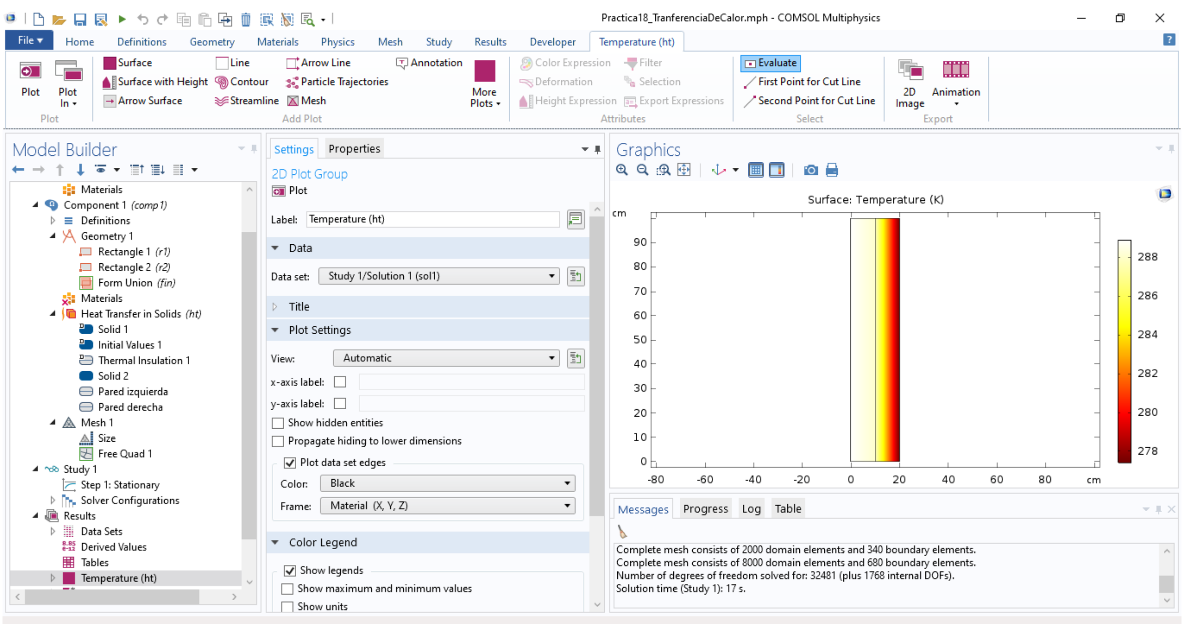


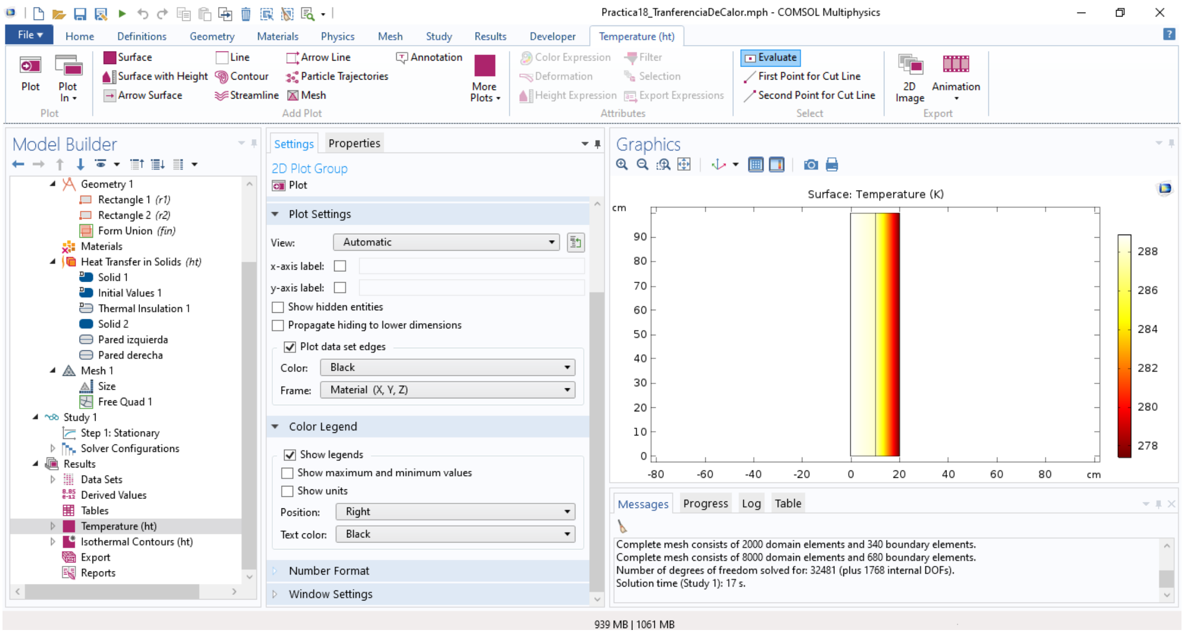
En custom le puedo poner la malla más fina.


Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Practica18_TranferenciaDeCalor.mph (root)
Global Definitions
Parameters
Materials
Component 1 (comp1)
Definitions
Geometry 1
Rectangle 1 (r1)
Rectangle 2 (r2)
Form Union (fin)
Materials
Heat Transfer in Solids (ht)
Solid 1
Initial Values 1
Thermal Insulation 1
Solid 2
Pared izquierda
Pared derecha
Mesh 1
Size
Free Quad 1
Study 1
Step 1: Stationary
Results
Settings
Size
Build Selected
Build All
Label: Size
Element Size
Calibrate for:
General physics
Predefined
Extremely fine
Custom
Element Size Parameters
Maximum element size:
1
cm
Minimum element size:
0.002
cm
Maximum element growth rate:
1.1
Curvature factor:
0.2
Resolution of narrow regions:
1
Graphics
Messages
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica18_TranferenciaDeCalor.mph
Saved file: C:\Users\diego\Documents\COMSOL\Practicas COMSOL\Practica18_TranferenciaDeCalor.mph
Formed union of 2 solid objects.
Finalized geometry has 2 domains, 7 boundaries, and 6 vertices.
Complete mesh consists of 2000 domain elements and 340 boundary elements.
883 MB | 1016 MB

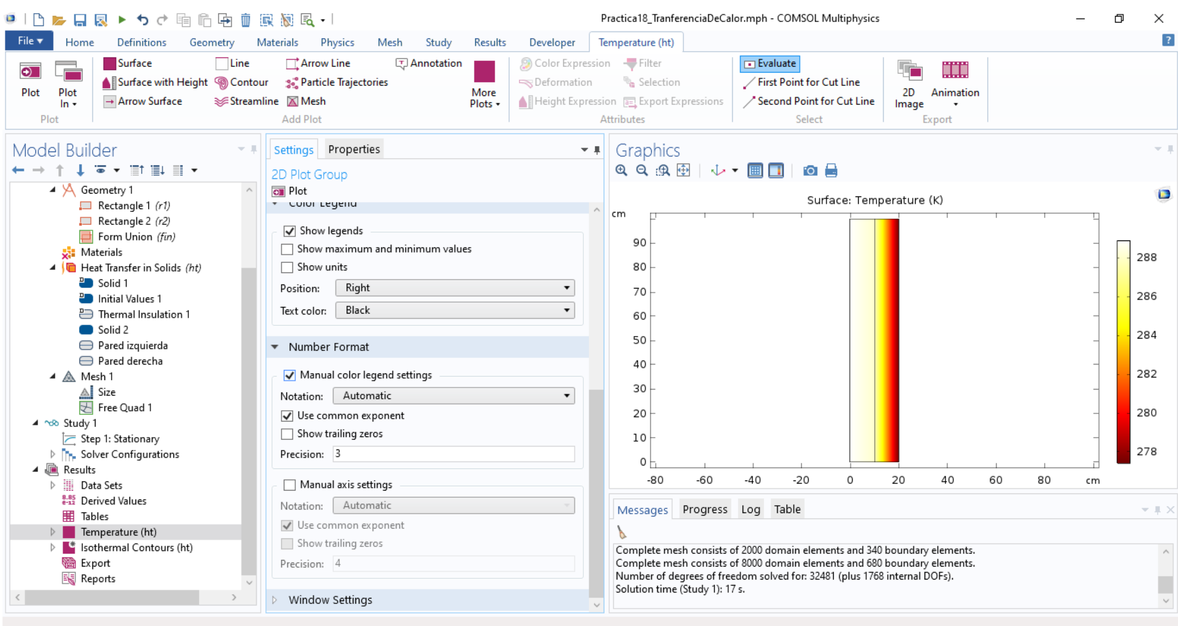

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings
Size
Build Selected
Build All
Build All (F8)
Label: Size
Element Size
Calibrate for:
General physics
Predefined
Extremely fine
Custom
Element Size Parameters
Maximum element size:
0.5
cm
Minimum element size:
0.002
cm
Maximum element growth rate:
1.1
Curvature factor:
0.2
Resolution of narrow regions:
1
Graphics
Messages
Formed union of 2 solid objects.
Finalized geometry has 2 domains, 7 boundaries, and 6 vertices.
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
881 MB | 1015 MB


Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Study
Compute
Study Settings
Generate default plots
Generate convergence plots
Store solution for all intermediate study steps
Information
Last computation time:
Graphics
Messages
Progress
Log
Table
Formed union of 2 solid objects.
Finalized geometry has 2 domains, 7 boundaries, and 6 vertices.
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
884 MB | 1016 MB

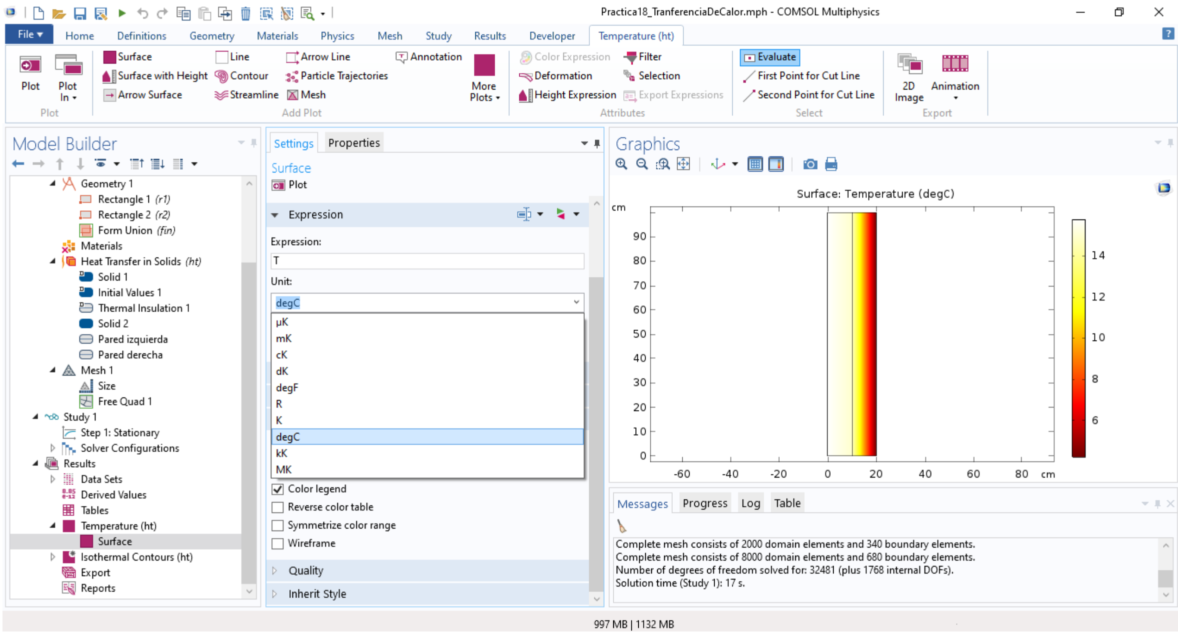

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
2D Plot Group
Plot
Label: Temperature (ht)
Data
Data set: Study 1/Solution 1 (sol1)
Plot Settings
Color Legend
Graphics
Surface: Temperature (K)
Messages
Progress
Log
Table
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
Number of degrees of freedom solved for: 32481 (plus 1768 internal DOFs).
Solution time (Study 1): 17 s.
939 MB | 1061 MB



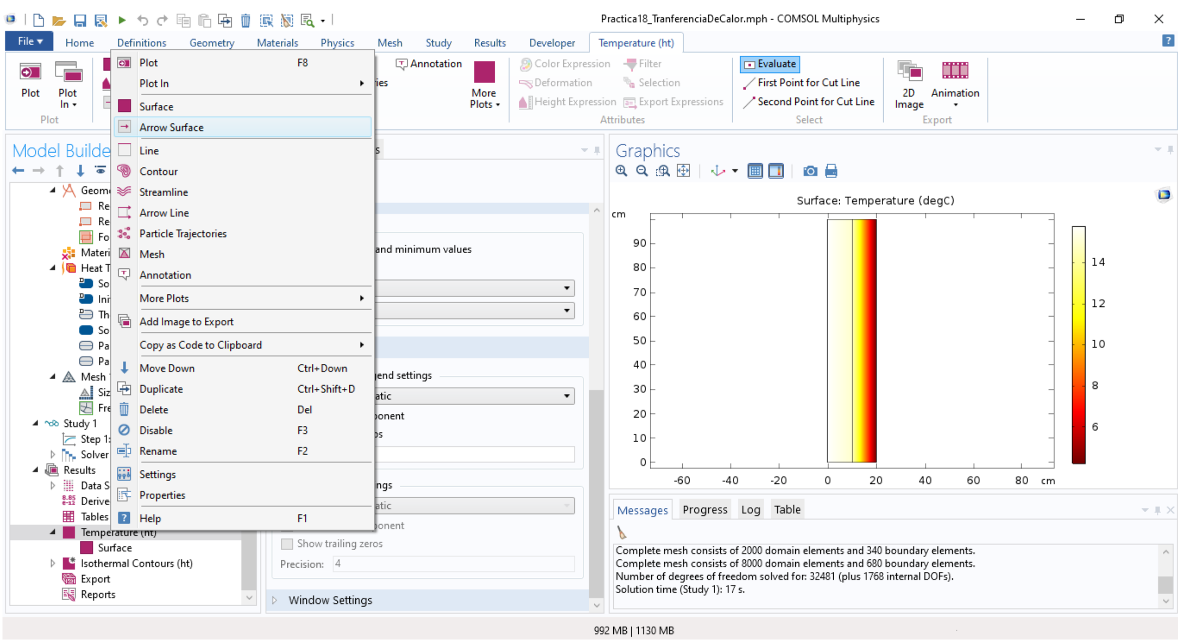
Aquí puedo hacer que el programa ponga más decimales en el resultado.



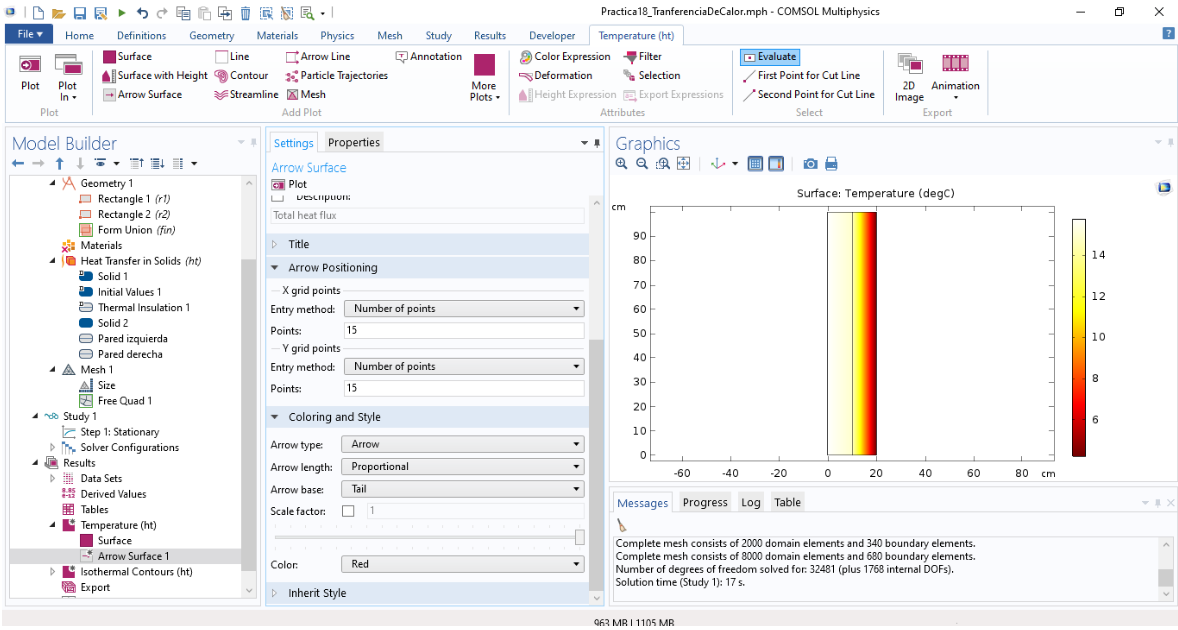
Igual lo puedo ver en grados centígrados.

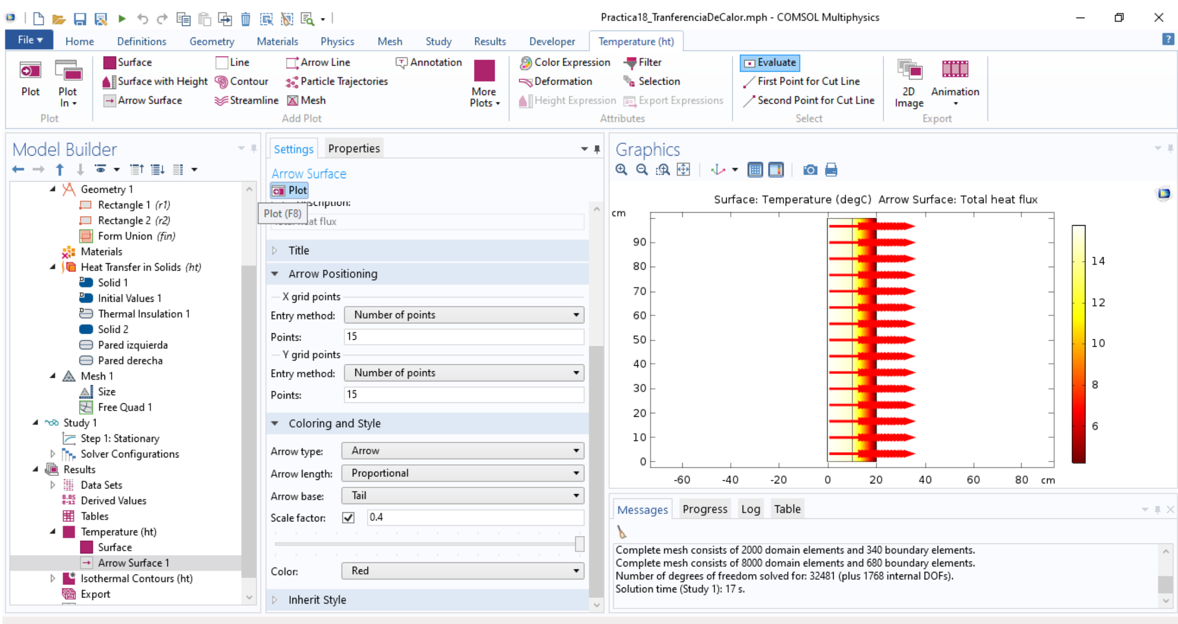


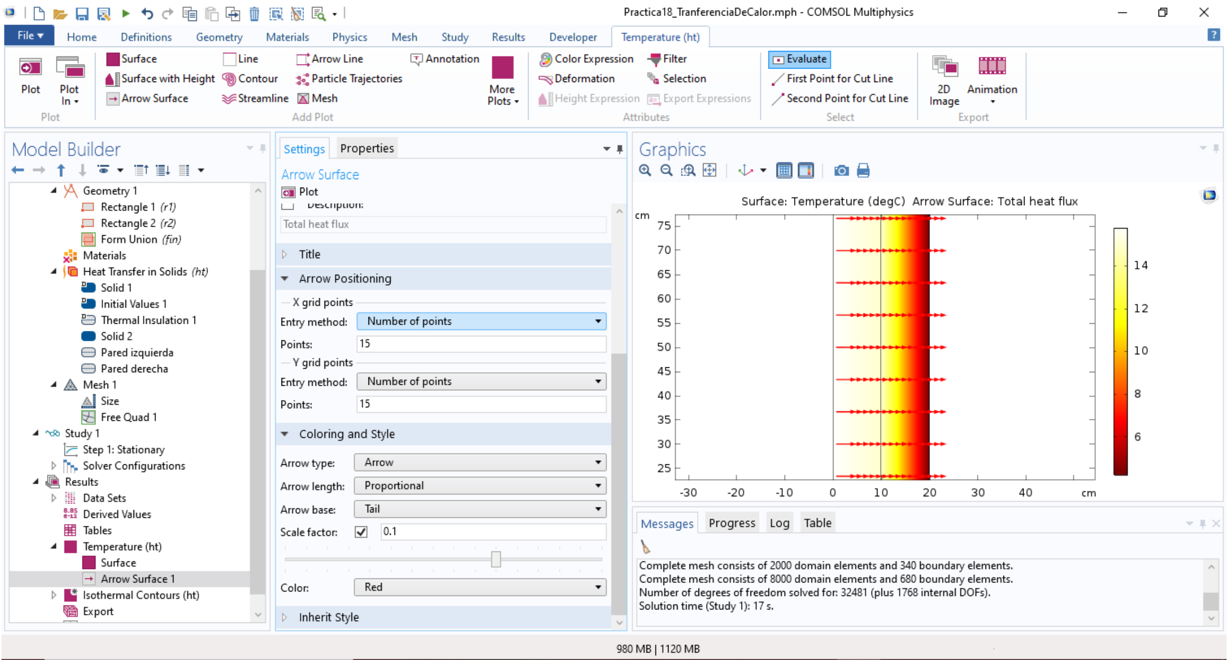
Si queremos ver el flujo de calor con flechitas hacemos lo siguiente:

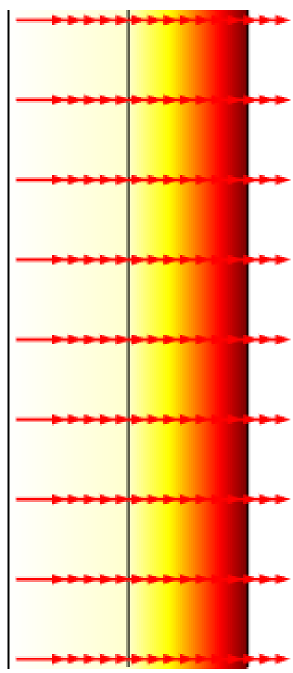


Puedo cambiar el scale factor para que haya más flechitas:

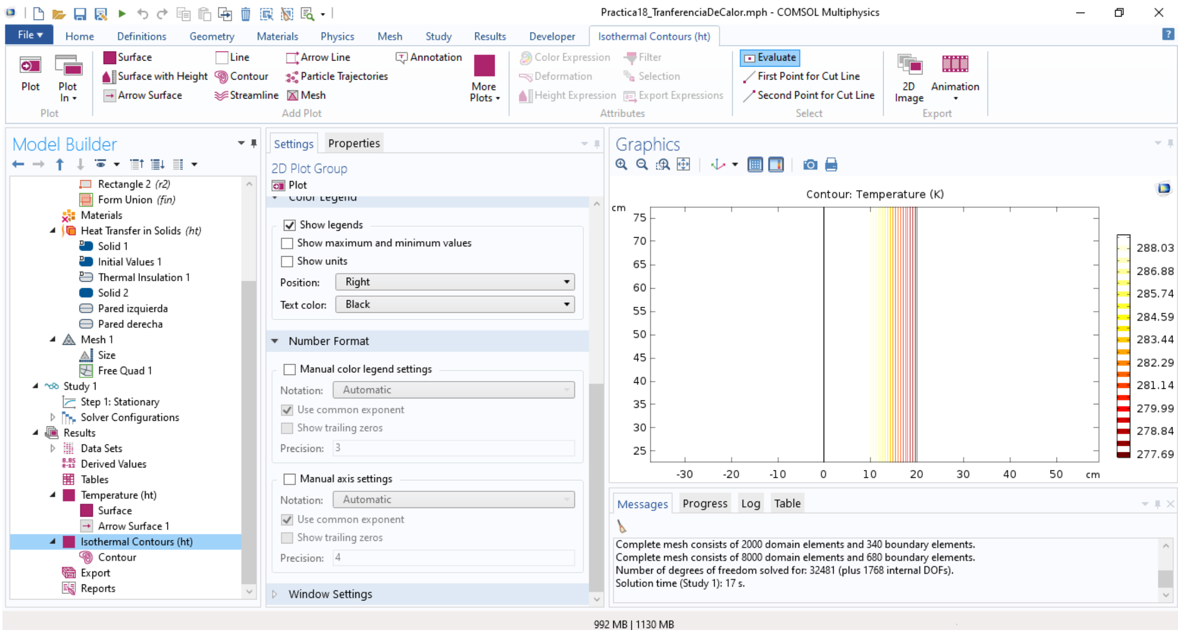

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Arrow Surface
Plot
Total heat flux
Title
Arrow Positioning
X grid points
Entry method: Number of points
Points: 15
Y grid points
Entry method: Number of points
Points: 15
Coloring and Style
Arrow type: Arrow
Arrow length: Proportional
Arrow base: Tail
Scale factor: 1
Color: Red
Inherit Style
Graphics
Surface: Temperature (degC)
Messages
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
Number of degrees of freedom solved for: 32481 (plus 1768 internal DOFs).
Solution time (Study 1): 17 s.

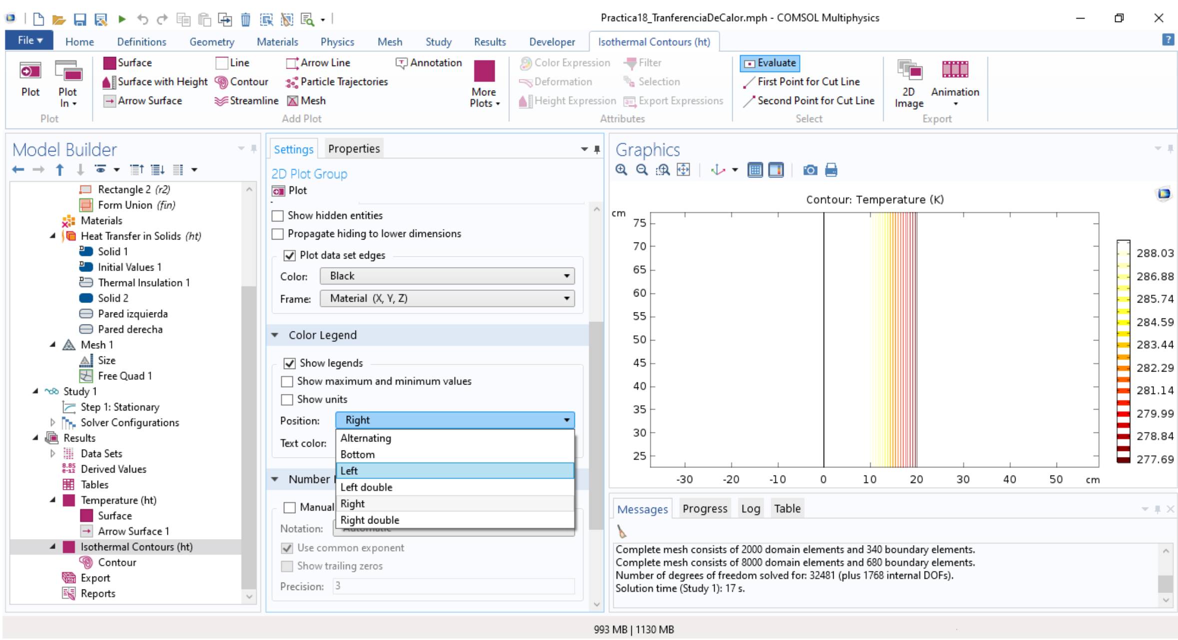

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings
Properties
Arrow Surface
Plot
Plot (F8)
Title
Arrow Positioning
X grid points
Entry method: Number of points
Points: 15
Y grid points
Entry method: Number of points
Points: 15
Coloring and Style
Arrow type: Arrow
Arrow length: Proportional
Arrow base: Tail
Scale factor: 0.4
Color: Red
Inherit Style
Graphics
Surface: Temperature (degC) Arrow Surface: Total heat flux
Messages
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
Number of degrees of freedom solved for: 32481 (plus 1768 internal DOFs).
Solution time (Study 1): 17 s.

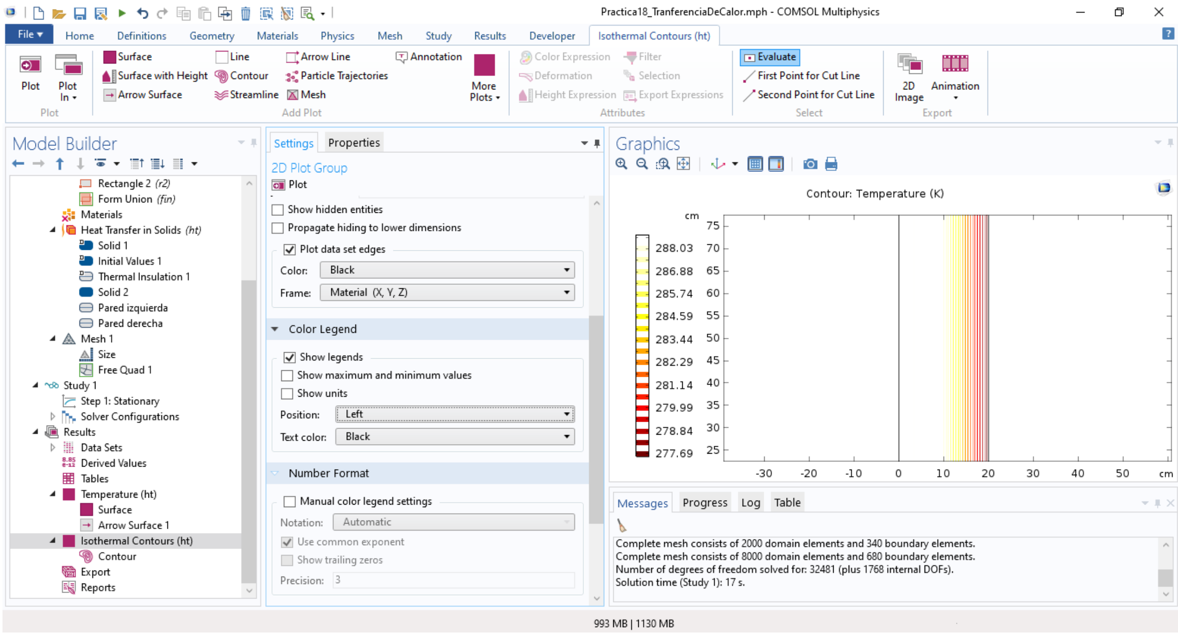




En isotermal contour igual veremos líneas que indican el cambio de temperatura:

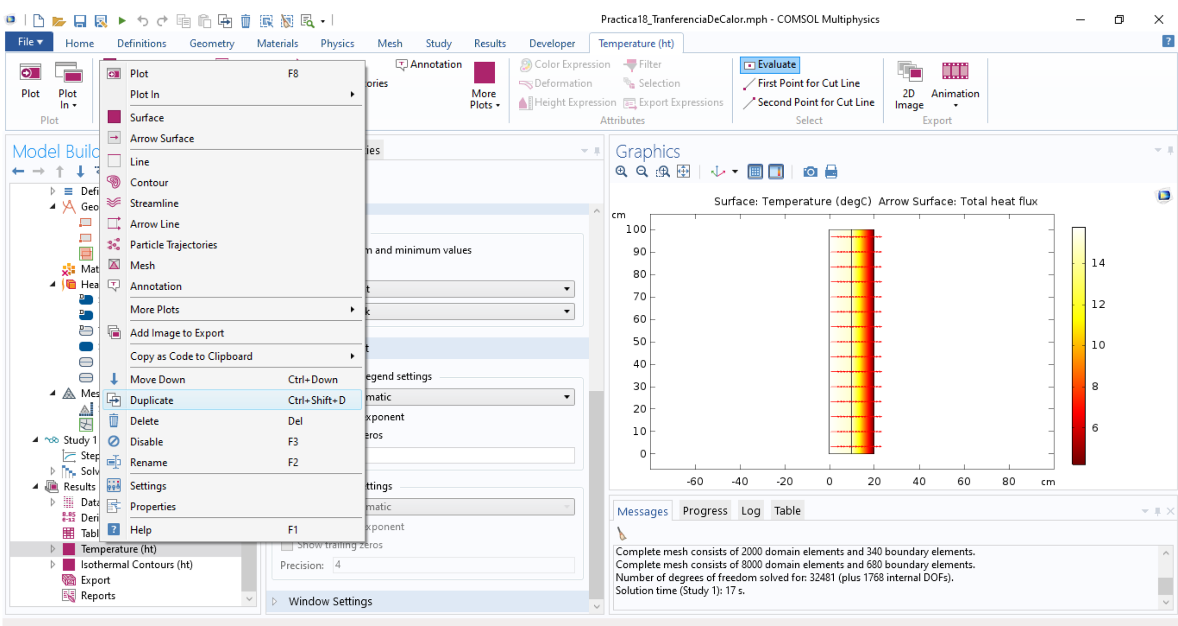

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
File Home Definitions Geometry Materials Physics Mesh Study Results Developer Isothermal Contours (ht)
Model Builder
Settings Properties
2D Plot Group
Color Legend
Show legends
Show maximum and minimum values
Show units
Position: Right
Text color: Black
Number Format
Manual color legend settings
Manual axis settings
Window Settings
Graphics
Contour: Temperature (K)
Messages Progress Log Table
Complete mesh consists of 2000 domain elements and 340 boundary elements.
Complete mesh consists of 8000 domain elements and 680 boundary elements.
Number of degrees of freedom solved for: 32481 (plus 1768 internal DOFs).
Solution time (Study 1): 17 s.
992 MB | 1130 MB

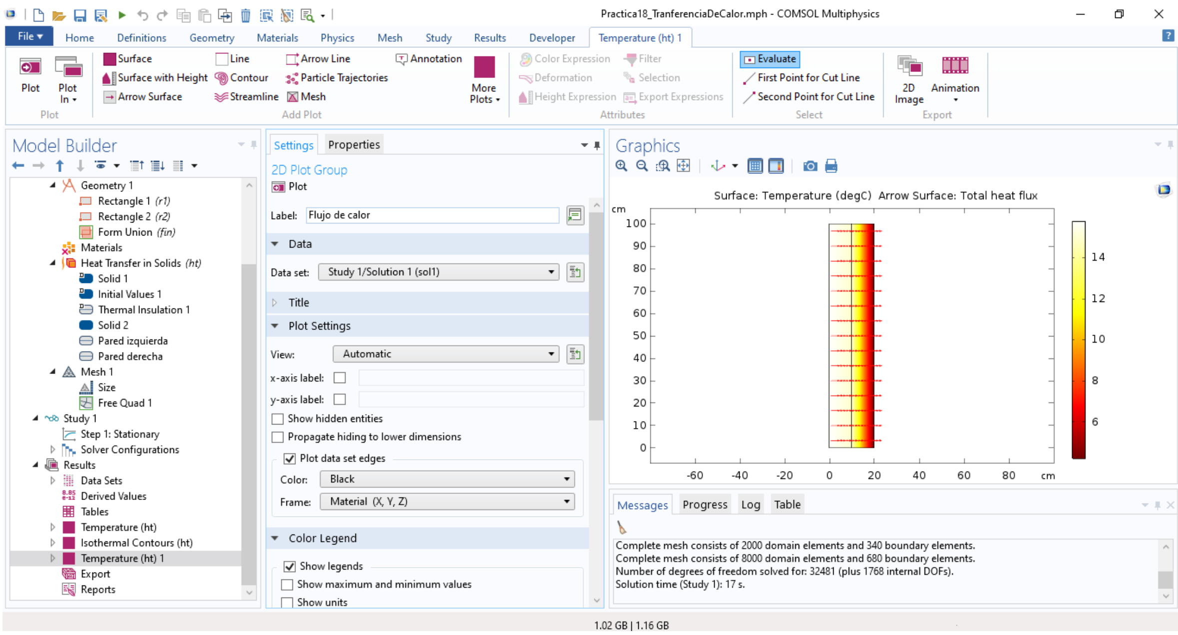

Practica18_TranferenciaDeCalor.mph - COMSOL Multiphysics
Model Builder
Settings Properties
2D Plot Group
Show hidden entities
Propagate hiding to lower dimensions
Plot data set edges
Color: Black
Frame: Material (X, Y, Z)
Color Legend
Show legends
Show maximum and minimum values
Show units
Position: Right
Alternating
Bottom
Left
Left double
Right
Right double
Graphics
Contour: Temperature (K)
993 MB | 1130 MB



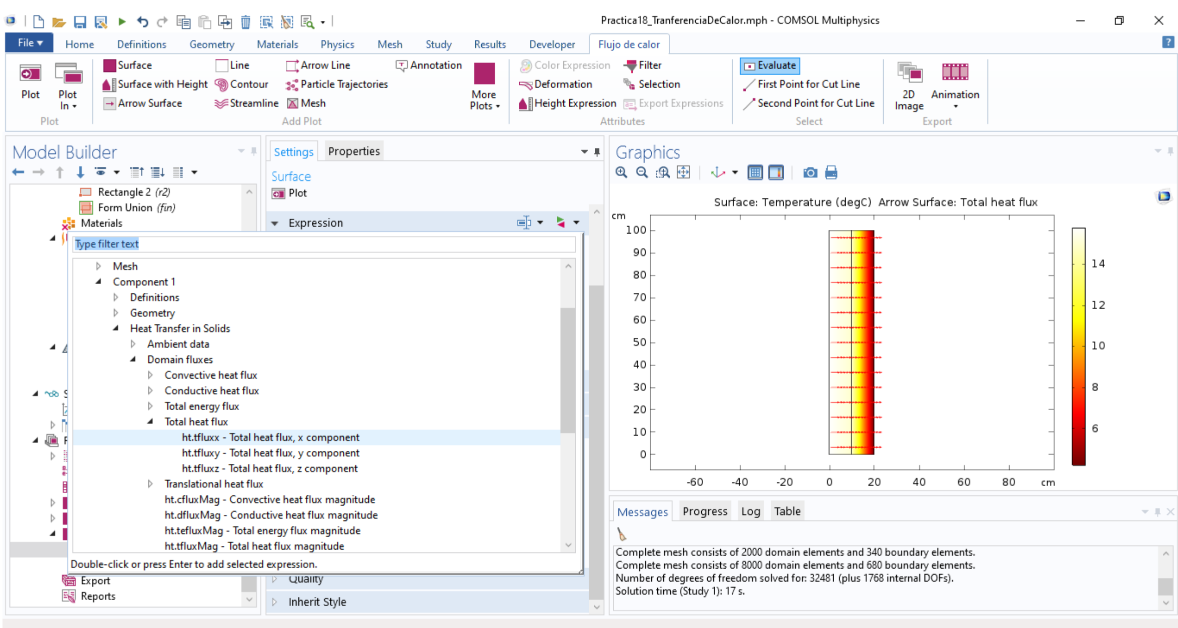
Esas se llaman isotermas y son líneas de temperatura constante.

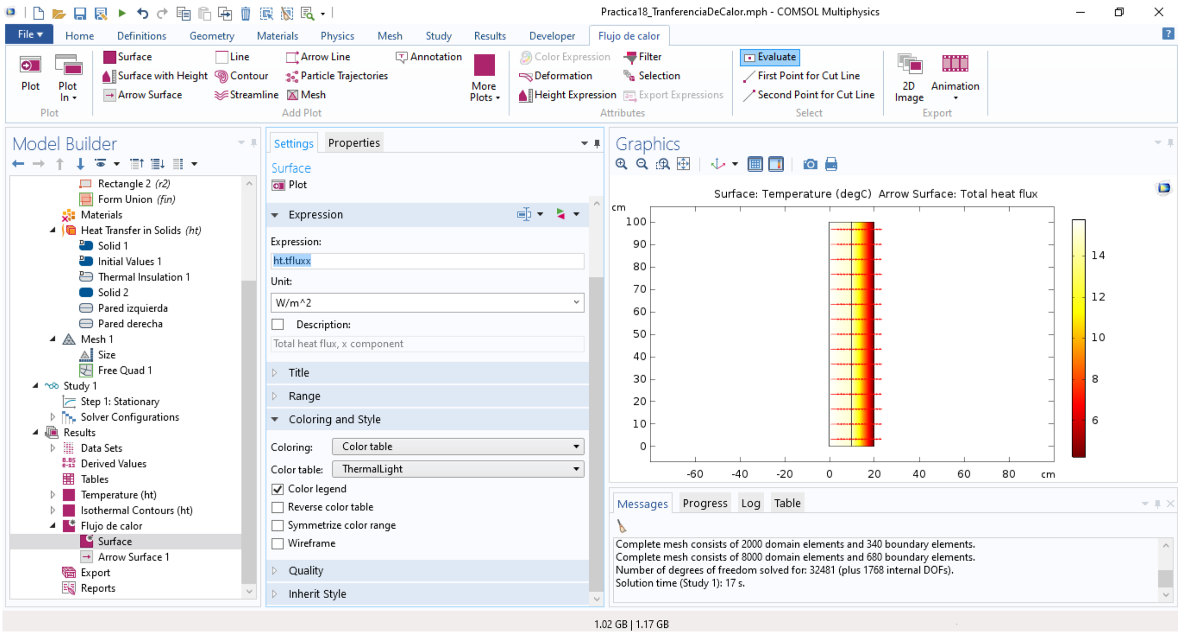
Si quiero ahora saber el flujo o la potencia de calor transmitida hago lo siguiente:

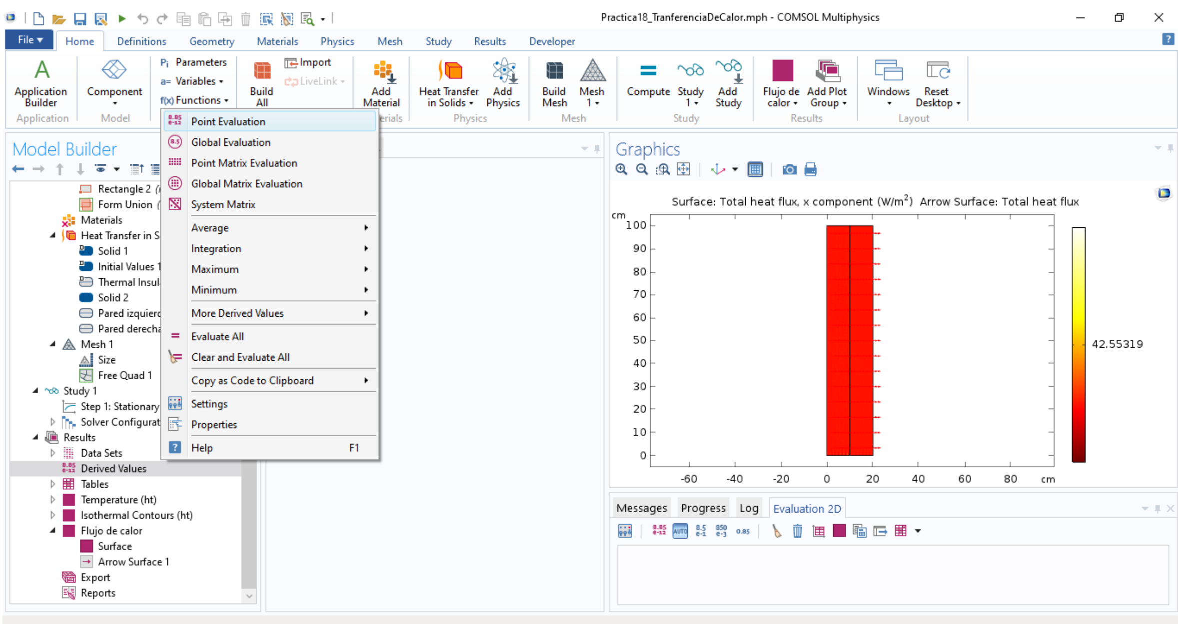




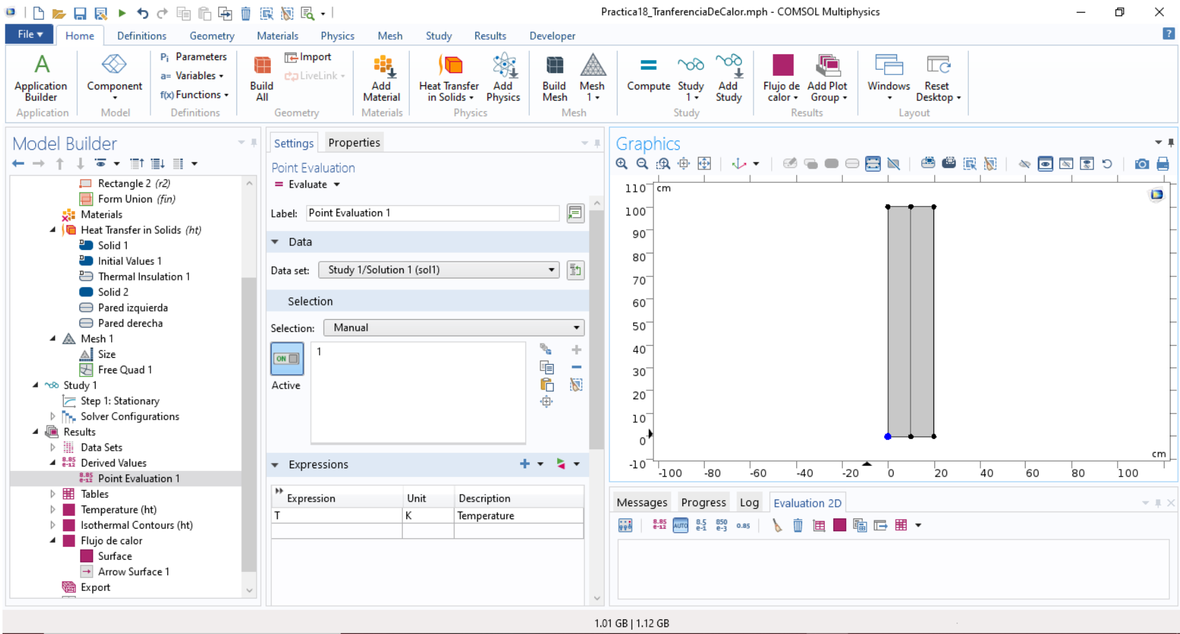
La expresión para calcular el flujo de calor es: ***ht.tfluxx***

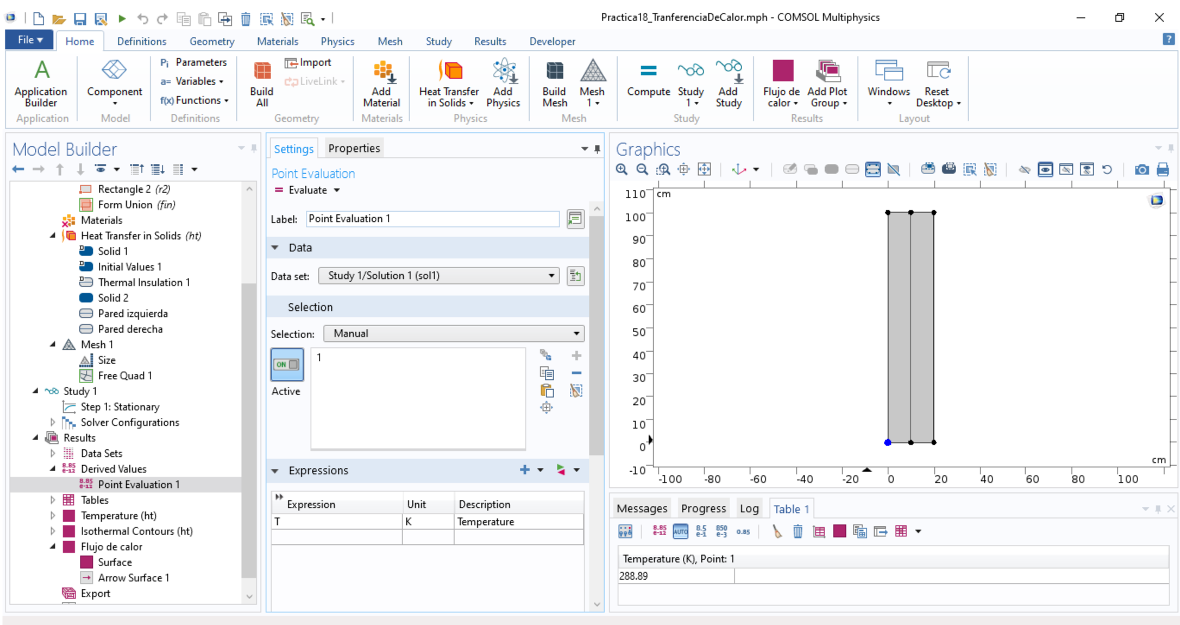






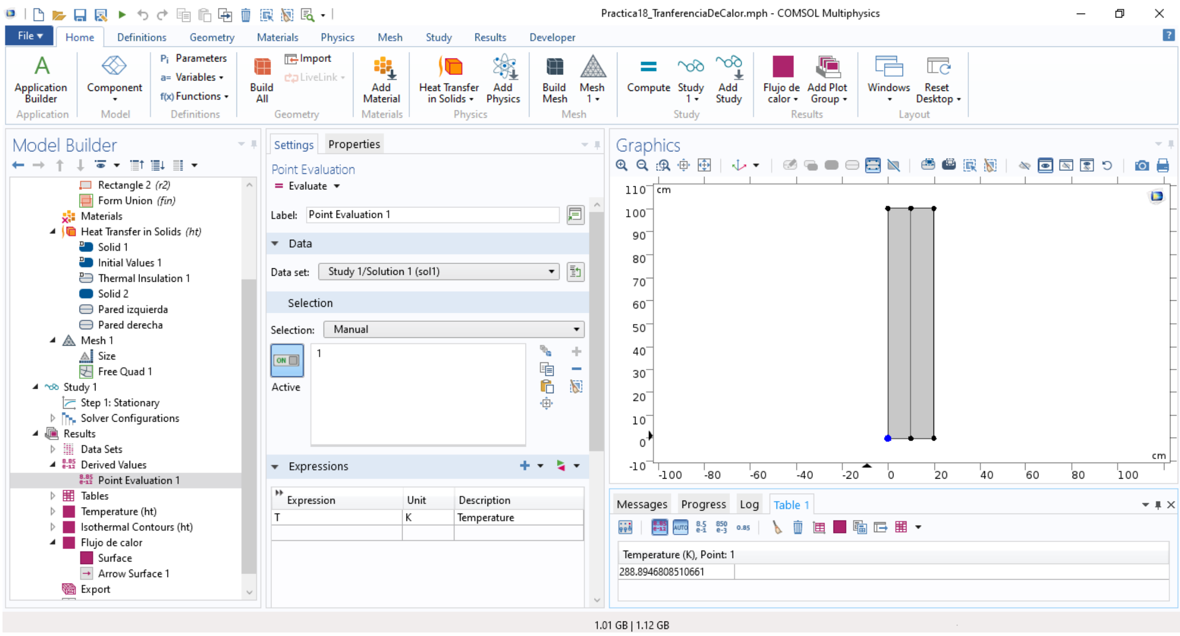
Ahora puedo ver la temperatura en las paredes de las placas.



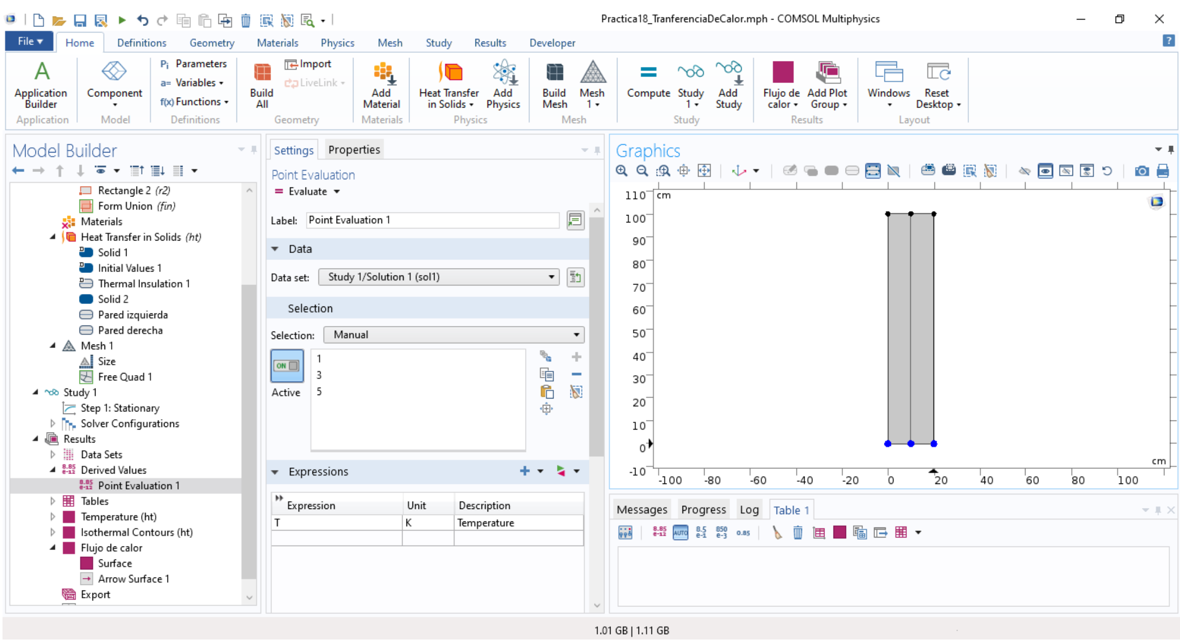


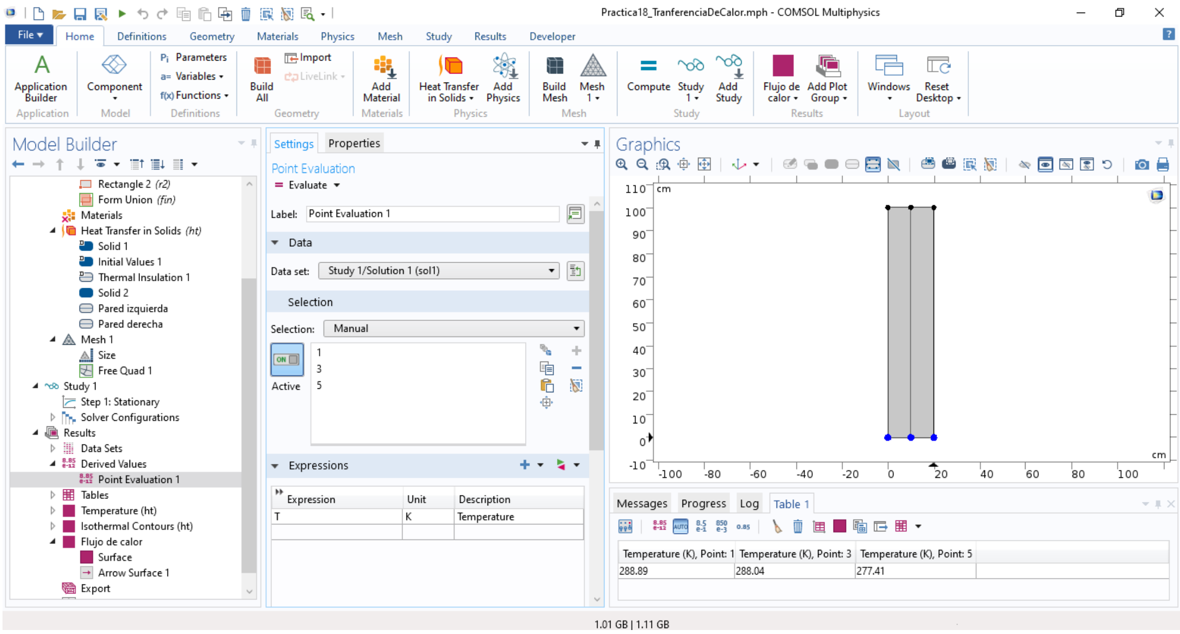
Me da un resultado donde Ti = 288.89.

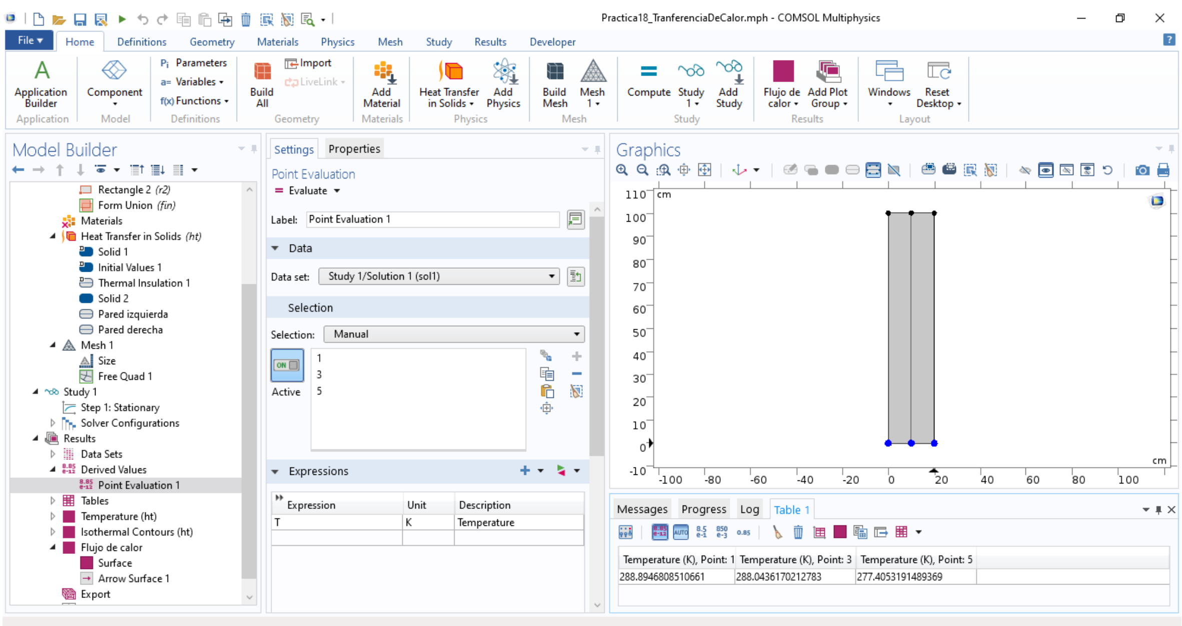
Con Full precition puedo poner más decimales.



Puedo hacerlo punto a punto o todo de jalón.







BIBLIOGRAFÍA:

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