The Unsteady State Field Equation

$$\rho C_p \frac{\partial \Phi}{\partial t} = \frac{\partial}{\partial x}(\kappa_x \frac{\partial \Phi}{\partial x}) + \frac{\partial}{\partial y}(\kappa_y \frac{\partial \Phi}{\partial y}) + \frac{\partial}{\partial z}(\kappa_z \frac{\partial \Phi}{\partial z}) + S$$

$\Phi = \Phi_1$ On surafce Γ_1

$$\kappa_x \frac{\partial \Phi}{\partial x} n_x + \kappa_y \frac{\partial \Phi}{\partial y} n_y + \kappa_z \frac{\partial \Phi}{\partial z} n_z = q \quad \text{On surface} \, \Gamma_2$$

$$\kappa_x \frac{\partial \Phi}{\partial x} n_x + \kappa_y \frac{\partial \Phi}{\partial y} n_y + \kappa_z \frac{\partial \Phi}{\partial z} n_z = h(\Phi - \Phi_\infty) \quad \text{On surface} \, \Gamma_3$$

Finite element equation:

$$\sum_{e}\sum_{j} K_j^e \Phi_j^{n+1} = \sum_{e} F_i^e$$

Depending on the definition of the variable Φ, the above field equation can be the governing equation of heat transfer, torsion of shafts, irrotational flow, groundwater seepage, electrostatic and magneto-static fields and fluid film lubrication.

Field equation input format

Currently the field equation can only be run by command mode.
Under esdc directory, type "command_line" to setup environment variables.
Then type "flusol' with .msh file as input to generate geo.* model.
Then rename geo.* model file as h1.da file. Here the h1.da is a finite element model file.
To run heat equation, change the solver = comp into solver = heat
Then type "flusol" with h1.da as input to solve the problem.

Procedure to generate heat equation model

Example h1 : Heat transfer between two concentric ellipses with fixed temperatures at 120 C and 35 C.

Example h2 : Heat transfer in square check box with different material properties

Example h3 : Square with heat convection

Example h4 : Square with heat flux

Procedure to generate heat equation model

1. Generate the .msh file for heat equation model. Must include the material ID in the .msh file

2. Under the *property cards specifies the thermal conductivity, density, cp. Such as : ckx,780.0, density, 4000.0, cp, 1280.0

3. Under the *material (solid propery card) lists the material corresponding to the material ID. Such as belows:

   **aluminum

   **material_id ,density, Cp, viscosity, ckx, cky,ckz. If only input ckx, then cky and ckz will

   be assumed as ckx=cky=cz

   1, 2700.0, 940.0, 0.0,204.0

   **glass

   **material_id ,density, Cp, viscosity, ckx, cky,ckz

   2, 2600.0, 830.0, 0.0, 0.81

BACK

FluSol : Field equation input format

To solve the field equation problems, following input are needed:

****begin** solver = Heat Equation
steady = on/off

***element connectivity** First card: **element number (I12), collector ID (I12), material ID (I12), wing ID(I12)** Second card: **element connectivity (8 I12)** example of first card: 100,3,1,1 example of second card: 1,21,22,2

***node coordinates**

Id, x, y, z

***material (material property)**

** material_id, density ρ, specific heat C_p, viscosity μ, thermal conductivities κ_x, κ_y, κ_z . If only gives the κ_x, then the program will assume $\kappa_x = \kappa_y = \kappa_z$

BACK

***initial**

N1,n2,inc, temperature

Where n1, n2 and inc are the begin, the end and the increment of node numbers.

***face (heat flux and convective heat transfer are applied on surface)**

N1, n2, inc, nface, type, heat flux/temperature, h-coefficient

Where nface is the element face number. Type=1, for the heat flux and type = 2, for the convective heat transfer of ambient.

For type=1, only heat flux value is needed.

For type = 2, ambient temperature and convective heat transfer coefficient h are needed.

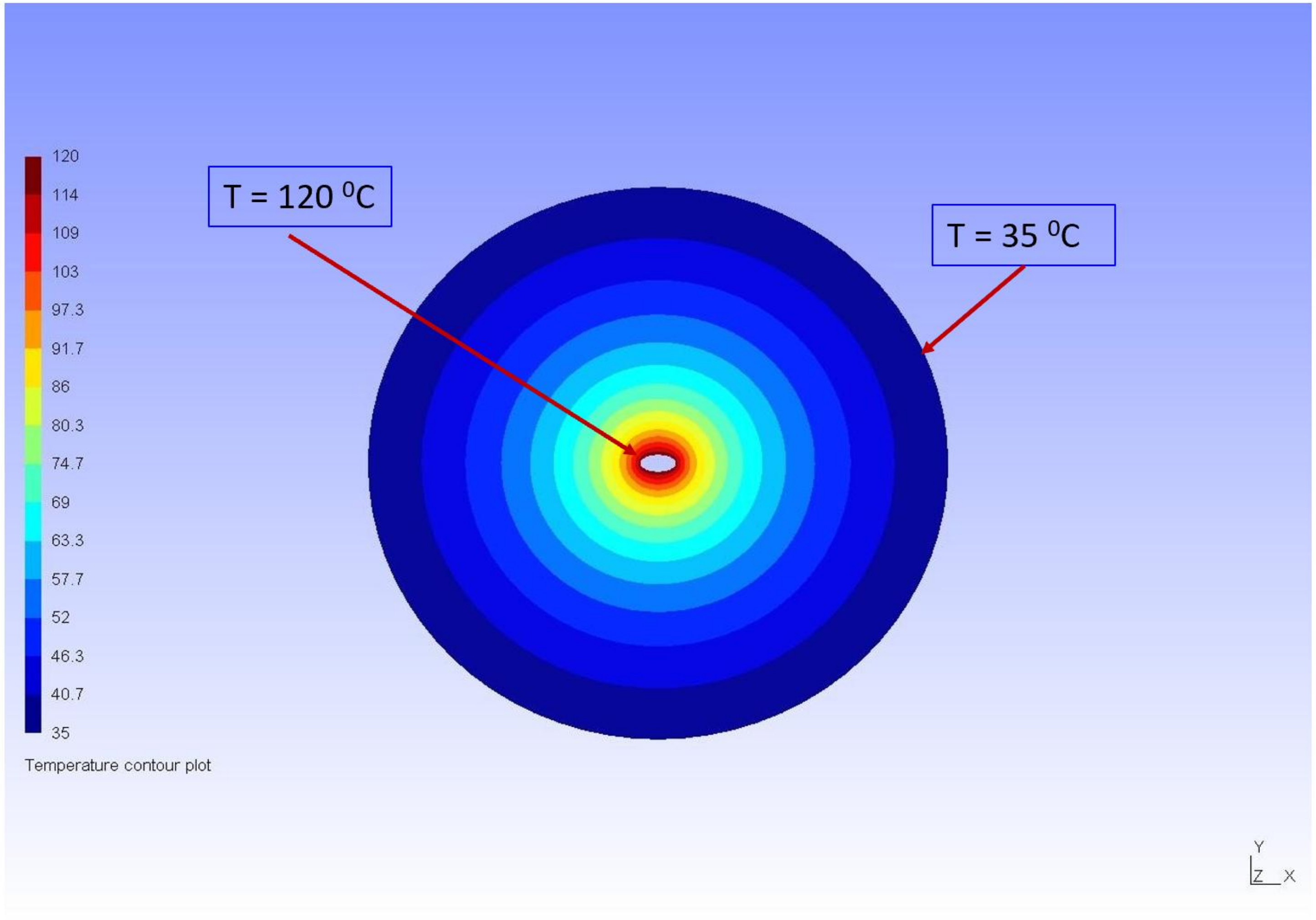
***boundary conditions (boundary conditions for temperature**

N1, n2, inc, temperature, value

***enddata**

BACK

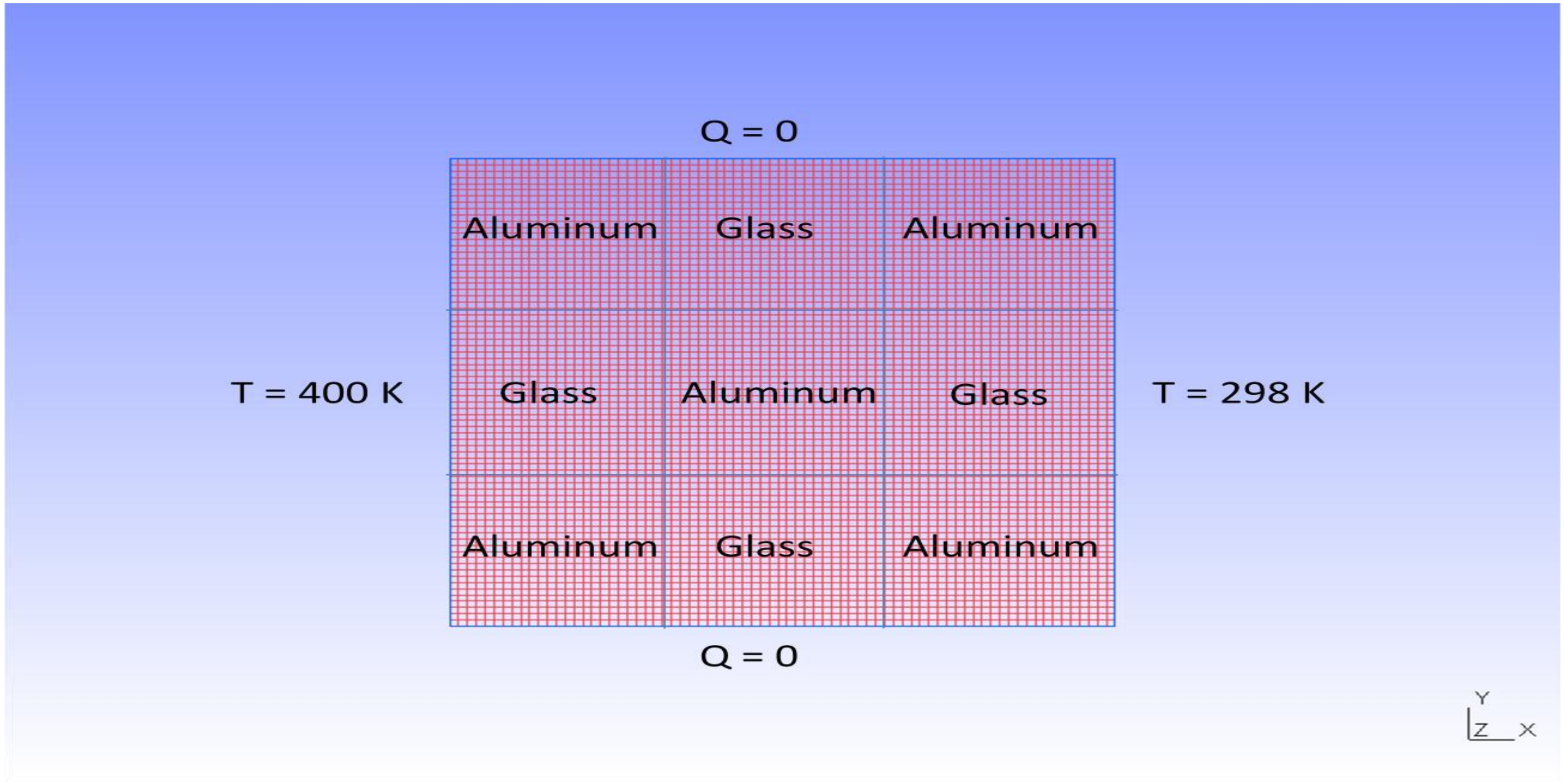
Example h1 : Heat transfer between two concentric ellipses with fixed temperatures at 120 C and 35 C.



BACK

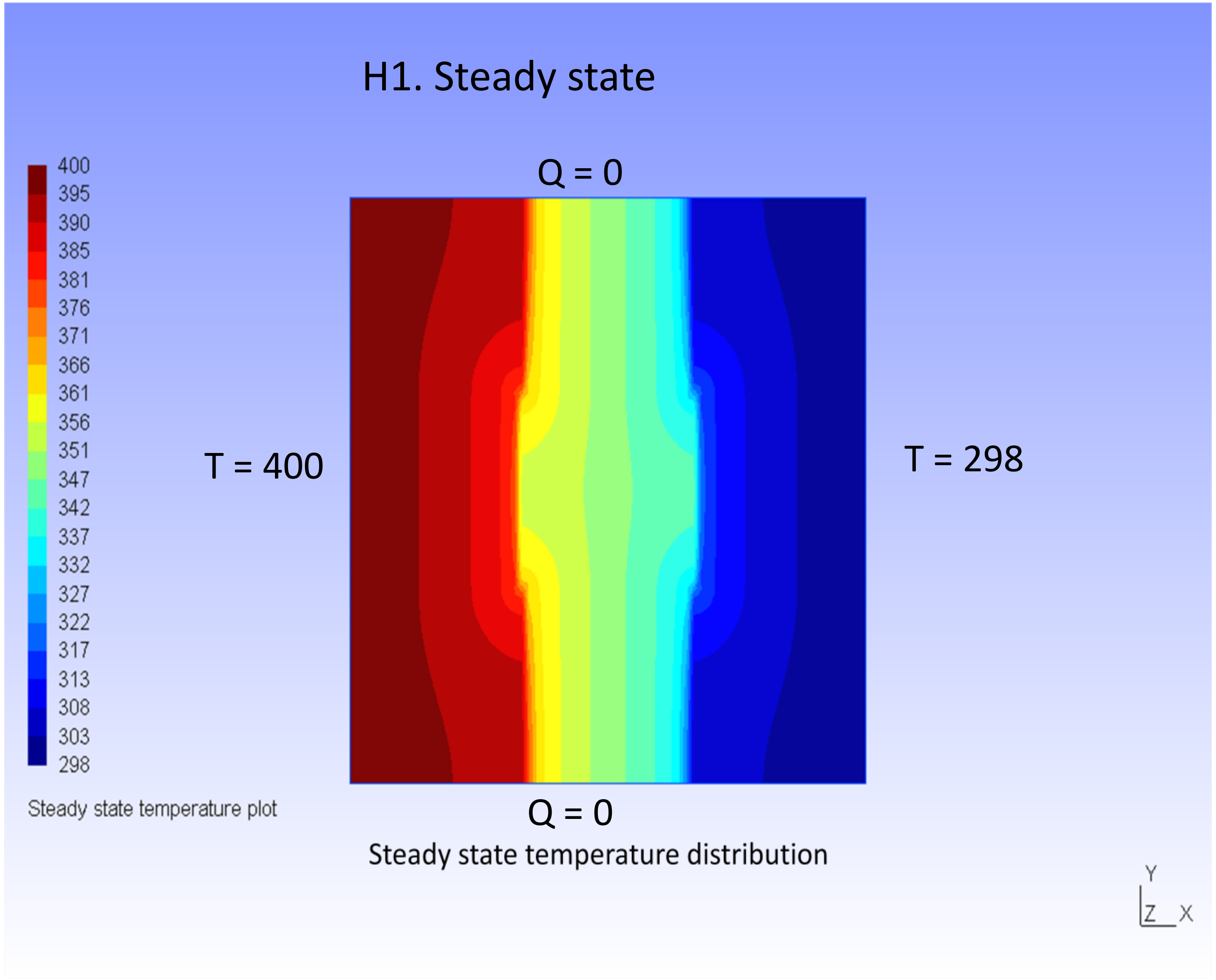
Fiure 1. Steady state temperature distribution between two concentric ellipses with fixed temperatures at 120 °C and 35 °C

Example (h1): Heat transfer in square check box with different material properties The material properties are applied as following:



BACK

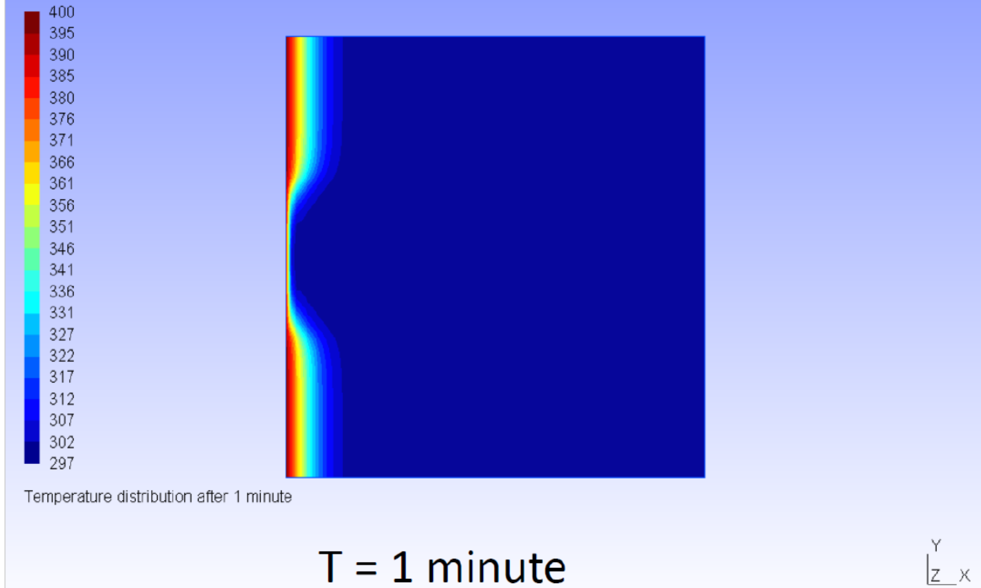
Figure 2. Computational domain for square check box.



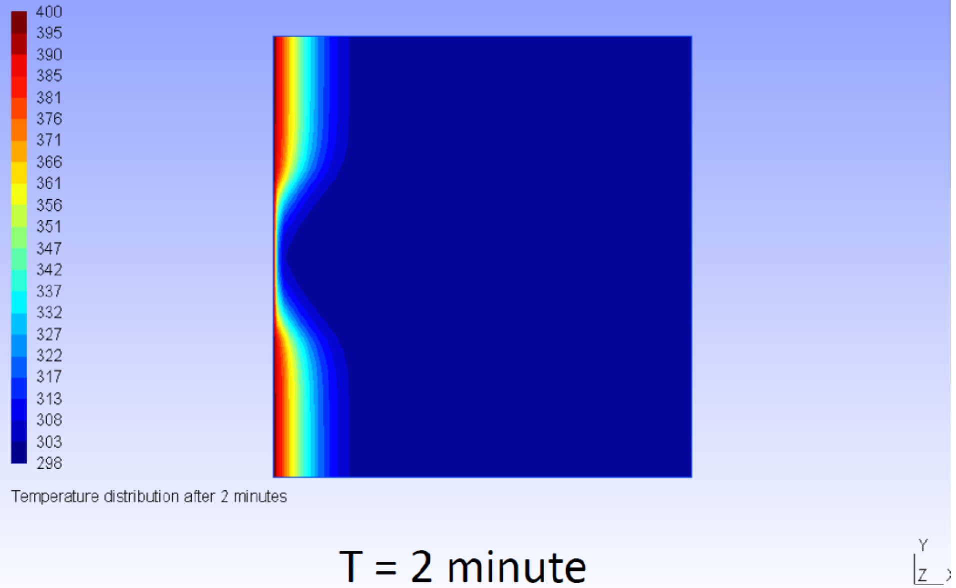
H1. Steady state
400
395
390
385
381
376
371
366
361
356
351
347
342
337
332
327
322
317
313
308
303
298
Steady state temperature plot
Q = 0
T = 400
T = 298
Q = 0
Steady state temperature distribution
Y
Z
X

BACK

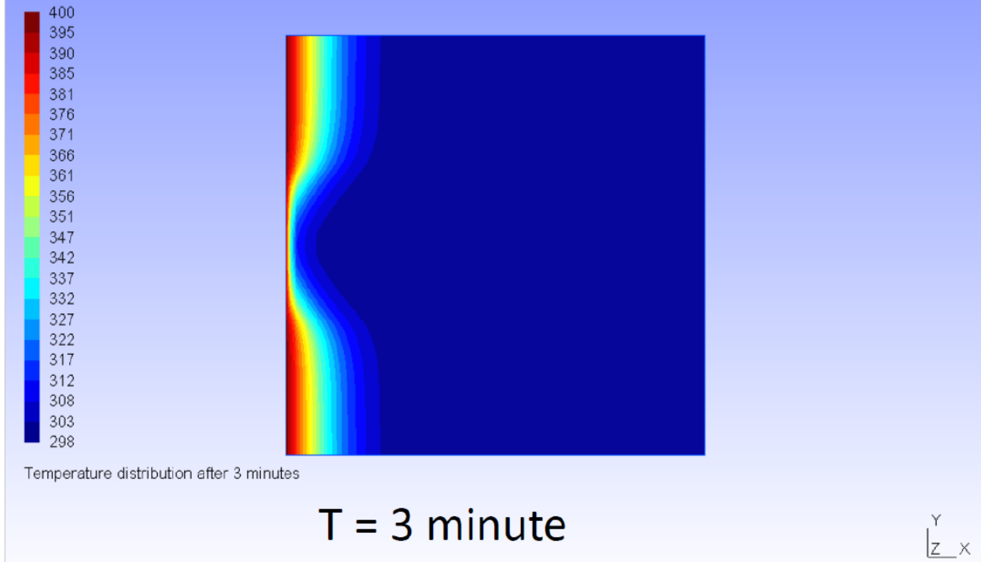
Example (h2): Transient simulations



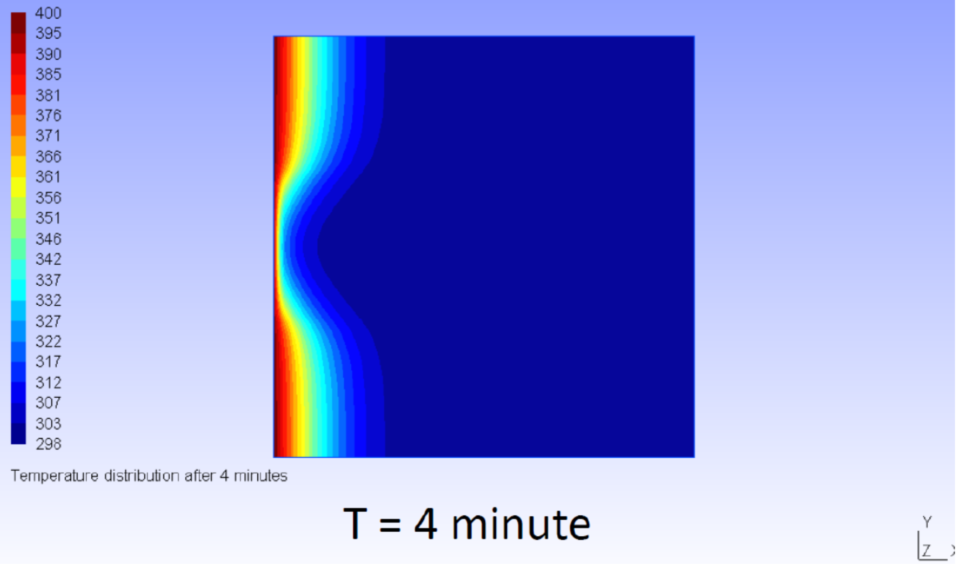
T = 1 minute



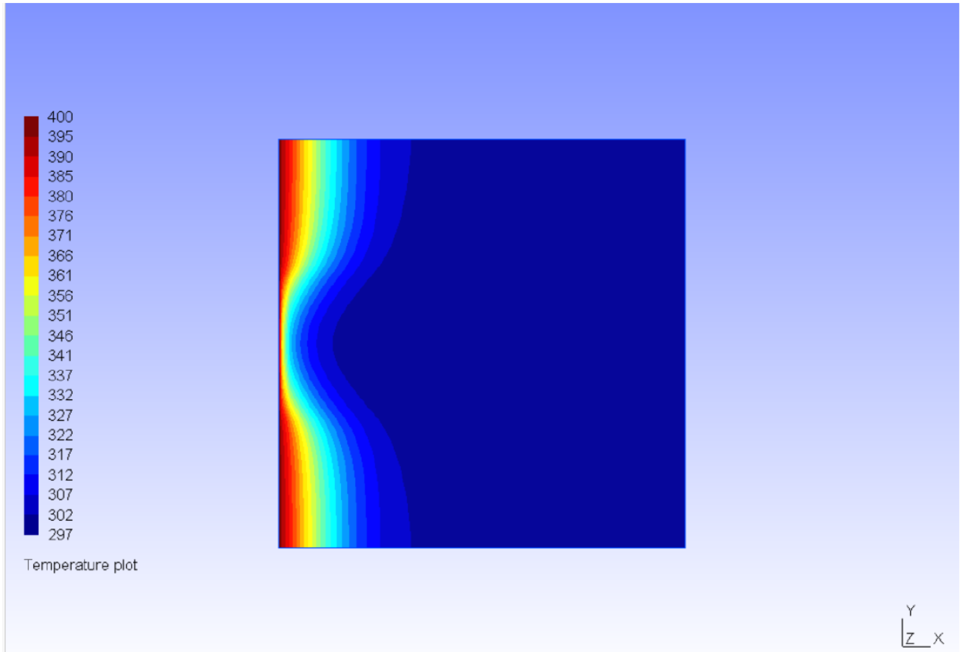
T = 2 minute



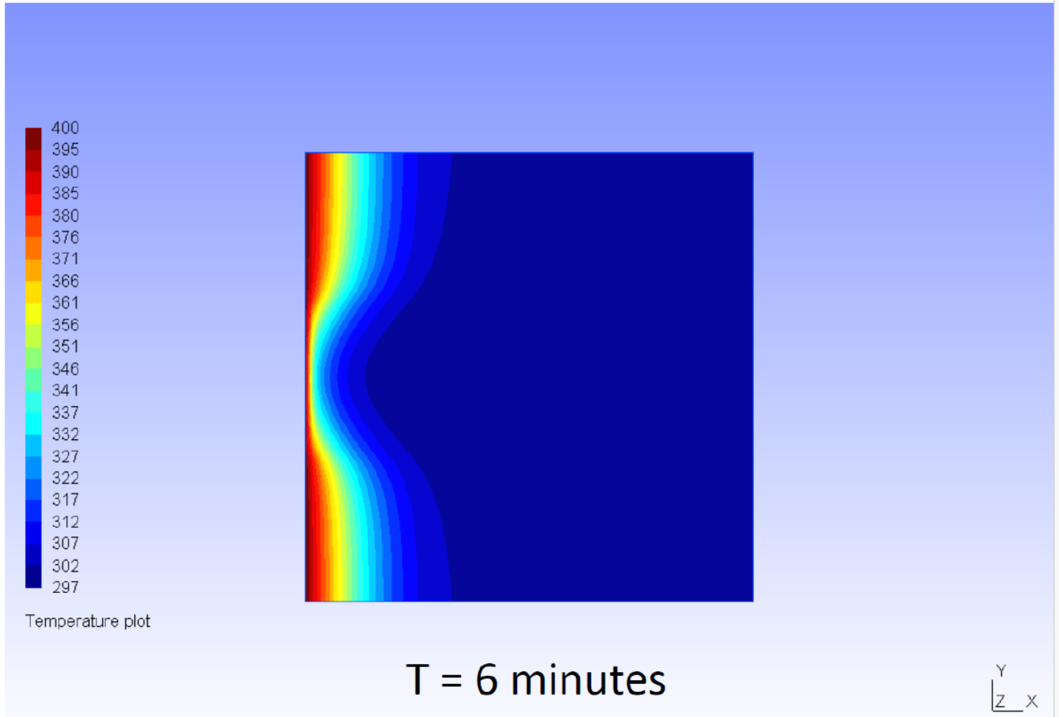
T = 3 minute



T = 4 minute

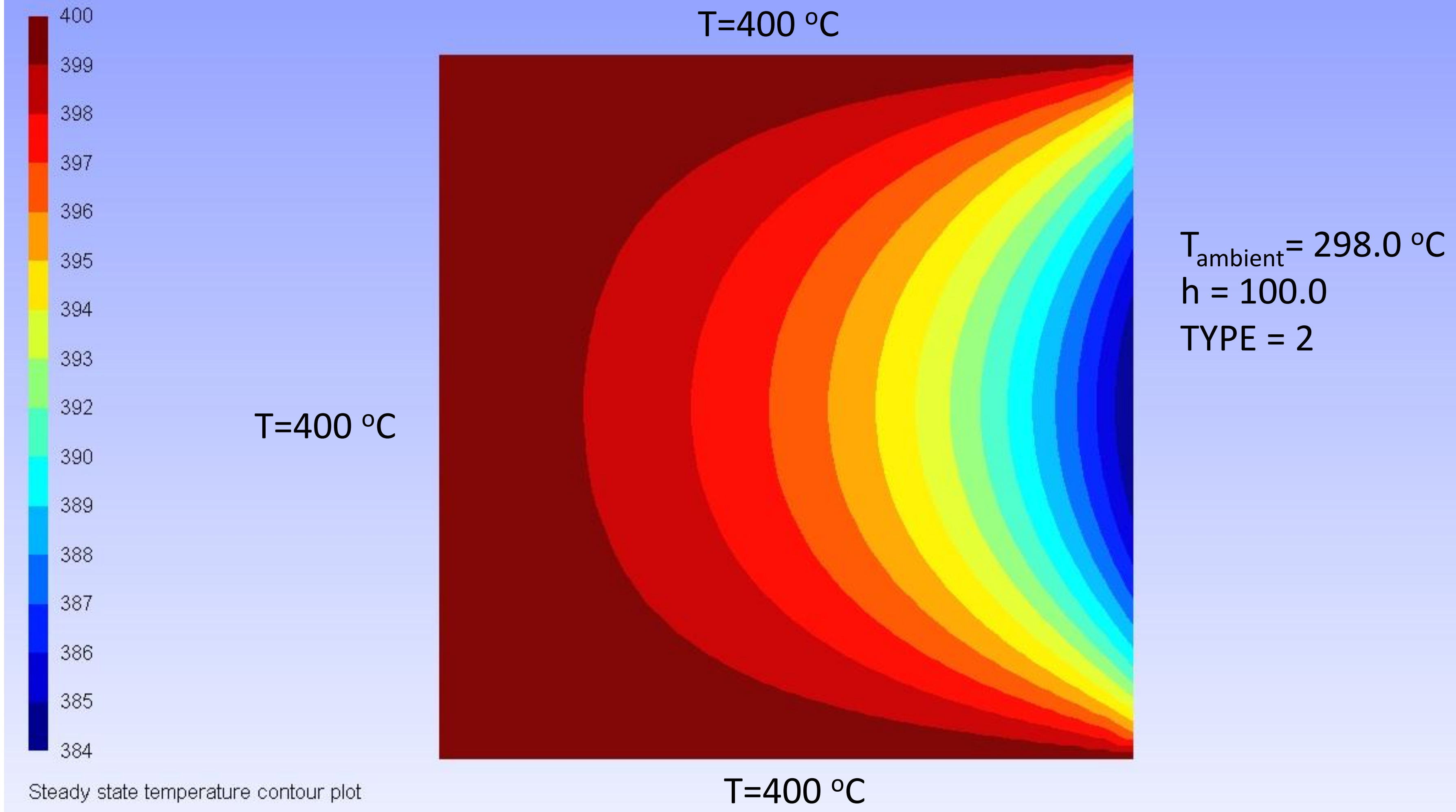


T = 5 minutes



T = 6 minutes

H3. Steady state convective heat transfer with $T_{ambient}$ = 298 K and h=100.0, TYPE=2



H4. Steady state with 3 side walls have T=400. and side 2 has heat flux = 10,000, TYPE=1