TUTORIAL #1

STEADY HEAT CONDUCTION 2D

Marek Jaszczur
TUTORIAL#1
Steady heat conduction 2D
- in rectangular domain

• Goal: Solution for 2D heat conduction problem
  1. Creating 2D simple geometry - DesignModeler
  2. Creating 2D mesh – Mesh
  3. Solver Set-up
  4. Results
  5. Plots
  6. Solution verification
Problem specification

• Flat plate with constant conductivity
• Consider 2D plate with height H and width W in cartesian coordinates x-y.
• Boundary conditions are presented below:

  • W=2 m ; H=1 m ; case (a) $k_{1(20^\circ C)}=$wood (0.173 W/mK)
  • $T_A=10^\circ C$ ; $T_B=100^\circ C$ ; $q_C=q_D=0$ W (adiabatic)
**Mathematical model**

- 2D heat conduction
- Steady, without generation
- $k=\text{const}$

\[
\frac{\partial^2 T(x, y)}{\partial x^2} + \frac{\partial^2 T(x, y)}{\partial y^2} = 0
\]

- Boundary condition I-type (Dirichlet)
  \[T(x, y) = T_A\]

- Boundary condition II-type (Neumman)
  \[-k \frac{\partial T(x, y)}{\partial x} = q_D = 0\]

- Boundary condition I-type (Dirichlet)
  \[T(x, y) = T_B\]

- Boundary condition II-type (Neumman)
  \[-k \frac{\partial T(x, y)}{\partial x} = q_C = 0\]
T1: Steady heat conduction 2D

Let’s go…..

• Open ANSYS Workbench

• Find WB Software in Menu Start --- Programs --- ANSYS 15.0 ----- Workbench 15.0
Steady problem can be solved using different modules from ANSYS WORKBENCH.

In this tutorial most advanced module for Fluid for Simulation is used.

**Fluid flow (FLUENT)**

(available in Anaysis Systems)

In order to select FLUENT module drag and drop or double-click.
1. For Geometry in Properties set-up
   Analysis Type \(\rightarrow\) 2D  !!!!
   If You leave default option 3D Your geometry will be three-dimensional

2. After 2D set-up double-click
   Geometry in order to start
   DESIGN MODELER
**T1: Steady heat conduction 2D**

**Geometry – DesignModeler**

1. Select **XYPlane** (just select and click) – as a working plane
2. Then select view at Face by clicking on icon
3. In tab **Sketching** are tools for plotting

*...at the beginning...*
At this step we will draw simple geometry – we will just plot and set-up dimension for rectangular.

In this version ANSYS 15 for the geometry DesignModeler, is used as a default module for geometry.

This tools allow us to create geometry from the beginning as well as to import geometry from other software for example from CAD software.

The first step is to check units!!!

In menu units check if You are working with meters (information about units can be also find in at rights bottom corner [METER]).

You can switch units in menu Units ------------------>

**PLANE**

Sketch (Plot) will be create at XYPlane to do that select To Look at Face select icon
At this step please select tab Sketching then Draw tools and Rectangle.

In order to have plot from (0,0) coordinates or attached to the axes select and enable constrains:

Last step is to draw rectangle from begining of XY-coordinates.

HELP - is something goes wrong --- use UNDO.
Now you should have rectangle but the size is probably wrong.

1. In tab **Dimensions** (use what is default GENERAL dimension)

2. Then select top side of rectangle and move cursor a bit top next do the same with left (or right) side

3. You should get new LABELS for example **V1** i **H1** for vertical and horisontal dimension
Dimensioning

- After dimensioning in Details View (left bottom corner) it is possible to set-up exact size for V1 and H2.

Don’t worry if your labels for Dimensions are not V1 and H2. When you create and delete new dimensions new numbers are used. The same for new Sketches.
Our Sketch READY but sketch can’t be used for computations !!! In order to performed simulations in ANSYS You need „BODY", and not a „SKETCH” !!!!.

• To create „BODY” we can use Sketch. In case of 2D body it will be just surface

• To create BODY select tab Modelling (not Sketching) then select from menu Concept > Surface From Sketches, as bellow:
Surface Body Creation

- Then select **XY Plane** and **Sketch 1**. (or other number)

- After Sketch 1 selection press Apply to accept selected Sketch in Details View **Details of SurfaceSk1**

- The last step is to find and click icon GENERATE

  Now Your BODY is ready to use in the next step.
T1: Steady heat conduction 2D

Surface Body Creation

If no ERROR we can enjoy with our PLATE-BODY (it should be with colour)

The thickness of our plate 2D is = 0 but You can change that
Body of 2D Plate is ready.

*However if You find any problem please download 2D_PLATE geometry from my web (2D_plate_geom file)*
T1: Steady heat conduction 2D

Surface Body Creation

- Our 2D PLATE (Surface Body) is ready
- Now you can close DesignModeler:
  menu File > close DesignModeler
- At this step You can save whole Project in Workbench menu
  File under easy name (for example plate_2d):
  menu File > save the project

- Next step is to create MESH no.3 (Mesh)

To RUN mesh module double-click 3. Mesh
Mesh

• At this step numerical mesh will be created. Mesh is required in order to performed computer simulations – because of methodology used

• Continuous space will be replaced by the discrete space

• Here the division is $Nx=20$ (length - $X$ direction) and $Ny=10$ (height – $Y$ direction) As a result 200 control volumes CV (elements) is created (21x11 nodes)
T1: Steady heat conduction 2D

Mesh - Uniform

One of the easiest mesh will be created, but this is sufficient for heat transfer problem.

To generate **Automatic mesh** just on mesh Tree RMB ans select **Generate Mesh**

Mesh is created however number of elements is not as desired (200)

-One of the method to control grid size is to move **relevance Slider** – click here and try

-To create new mesh press icon **generate mesh**
After selecting or changing any mesh parameters
Press
generate mesh
Mesh – Edge Sizing

• Mesh looks good however number of elements is not Nx=20, Ny=10.

To divide rectangle side into specified number of elements select rectangle edge:
Press edge filter

Select rectangle edge

RMB > Insert > Sizing
Mesh – Edge Sizing

- After choice sizing the selected – edge is taken into account

- For this edge in window Details of „Edge Sizing“ several settings can be applied
  - for example number of division
  - then type number of =10

- Repeat this for all edges

You can select more than one edge using <Ctrl> + (click)
Mesh – Edge Sizing

- After selecting for all 4 edges desired number of division,
- press **generate mesh** icon. Finally the mesh can looks like bellow:

**Warning**! If generated mesh is not 20x10=200 elements it doesn’t mean that any mistakes has been done. This is because for option **Behaviour**= (see previous slide) the selection is **SOFT**. Selection Soft mean that computer do not have take our setting very strictly rather as guide. To change the above change **SOFT** into **HARD** – now mesh have to be 20x10!!
With HARD selection mesh may looks as bellow:

- Edges are divided correctly but the plate surface (internal part – body) is not divides as we wish
- To change that go to the next slide
To create uniform structural mesh:
Press **Face filter**

Tap rectangle surface

**RMB > Insert > Mapped Face Meshing**

Click on **Generate Mesh icon**
Now mesh is ready

In Mesh details window you can see mesh size (nodes, elements)

200 Elements
Before we will proceed next step and go to the solver it is very useful to give names for all edges. This allows easy recognition them in Fluent solver.

- Select edge filter
- Select any edge then:
  RMB > Create Name Selection

Then type desired name
Repeat procedure for all edges
- Top, Bottom, Left and Right
It is also possible to give name for whole body (plate)

Close meshing and back to Workbench
MESH for 2D Plate is ready.

However if You find any problem please download 2D_PLATE mesh file from my web (2D_plate_mesh file)
Before proceed to the next step **Setup mesh update is required**. To do that RMB and **update**

- symbol should change from ⚡ into ✓

Then You can go to the next step **SETUP**
When you click on Setup, FLUENT solver will run.

Welcome window will appear with few settings:
- Dimension (here because of geometry is 2D)
- Double precision (please enable)
- Serial/Parallel computation (leave default) 
  (each CPU may require license !!)
- Then proceed OK.
This is default solver Fluent v15 window (with plate)

Selection tree

Graphical window

Text window

You can also type here
First step is to select General

- Check Settings:
  - Steady
  - Planar

- Then go to the next step Models
At this step enable Energy model and go to the next step Materials.
Select
Create/Edit...

To select new material wood proceed:

- Fluent database
- Material type – solid
- In Material list find wood

Then COPY and CLOSE and CLOSE next window
At present in the list of available materials for Solid materials wood appears as possible choice with new material we can go to the last settings step: Boundary Conditions
At this step boundary condition have to be set for all boundaries

- select desired boundaries for example **left**
- then press **edit**
- switch **Thermal tab**
- and type required conditions (for Left 10 C i.e. 283 K)

Remember to repeat procedure for **ALL BOUNDARIES** select proper conditions
• We have taken for database wood material but up to now it is not taken into consideration

• To set-up material wood:
  - go to **Cell Zone Condition**
  - select **plate** (or any other name object if you don’t give name plate)
  - switch **Type** to **solid** and type OK.
  - In next window select Material Name **wood** as material for plate
The last step is to Run Calculation.

Set-up (maximum) **number of iteration 10**

..... and **Calculate** (in case of initialisation question click OK)

If no error calculation should iterate and **Residuals** plot should appears.
T1: Steady heat conduction 2D

Fluent – Plot solution

To Plot solution for example temperature Field:
- go to the Graphics and Animations
- In Graphics window double-click Contours
- In Contours of select Temperature
- In Options enable Filled then Display to see Plot

Your solution is READY
More detailed information can be obtained from simple X-Y plot. To create plot of temperature distribution along top side proceed as follows:

- go to Plot in Fluent tree
- in Plots window select XY Plot
- in new Solution XY Plot windows select temperature and top surface
- new plot should appears
To write presented results into text file just recall last window and enable **write to file**.

- Now instead of plot results are save into text file:
- Give name **T.dat** and try to open this file with **notepad** or import to **Excel**.
THE END
Appendix #1

variable thermal conductivity

\[ k = \text{var} \]
In Fluent tree select **Materials** then in **Materials** window all Fluid and Solid materials can be seen.

In order to modify material properties double-click on wood.

As can be seen thermal conductivity for wood is 0.173 but this value can be changed by typing a different value.

There is also the possibility to select different from constant properties.
scroll thermal conductivity menu and change **constant** into **piecewise-linear**

new window appear. Set-up two points:
1. for 273 K  $k=0.1$

2. for 373 K  $k=1.5$

After selection repeat calculation and draw XY plot
For non-constant conductivity temperature distribution is non-linear

1. for 273 K  \( k=0.1 \)
2. for 373 K  \( k=1.5 \)
Appendix #2 boundary conditions of different type
T1: Steady heat conduction 2D

Different B.C.

1. Set up constant conductivity $k$
2. Change B.C. as in picture
3. In Fluent tree

Boundary conditions
than in tab Thermal

4. Repeat calculation
T1: Steady heat conduction 2D

Different B.C.

- change top B.C into Temperature $T=100K$
T1: Steady heat conduction 2D

Different B.C.

- change top B.C into convection
- heat transf coef = 1 W/m²K
- free stream temp. = 100K
Different B.C.
- change top B.C into Radiation
  - emissivity=1
  - external temp.=100K

$T_{\text{free}} = 100\,\text{K}$, $\alpha = 1\,\text{W/m}^2\text{K}$

Wood plate

adiabatic
T1: Steady heat conduction 2D

Different B.C.

Change top B.C. >> **Mixed**
-emissivity = 1
-external temp. = 100K
-heat transf coef = 1 W/m²K
-free stream temp. = 100K

\[ T_e = 100K \quad \varepsilon = 1 \]

**not this**
Appendix #3
internal heat generation
\[ q_v = 10^6 \, \text{W/m}^3 \]
Problem specification – with heat generation

- Flat plate with constant conductivity
- Consider 2D plate with height H and width W in cartesian coordinates x-y.
- Boundary conditions are presented below

- W=2 m ; H=1 m ; case (a) \( k_{1(20^\circ C)} = \) wood (0.173 W/mK)
- \( T_A = 100^\circ C \); \( T_B = 100^\circ C \); \( q_C = q_D = 0 \) W (adiabatic)
Problem specification – with heat generation

- Set-up boundary conditions as on problem specification
- In Fluent tree select **Cell Zone Conditions** and Edit Plate (or any different name) Zone
- enable Source Terms
- go to the tab Source Terms
- Edit Energy sources:
  - set-up number of sources = 1
  - select constant
  - put value 1000 W/m³
T1: Steady heat conduction 2D

Problem specification – with heat generation

- Run Calculation
- Plot Contour for Temperature
- Check max/min Temperature
Problem specification – with heat generation

- Create XY-plot for Temperature at Top line:
  - go to Fluent tree and select Plots
  - in Plots Window select XYPlot and double-click
  - set-up Temperature to display
  - select Top Line to Plot

- Plot presented bellow should appear after pressing Plot button
Problem specification – with heat generation

• Write calculated results to the text file but selecting [write to File] option

• Save Temperature data into T_gen.dat file and compare at home with analytical solution.

• for presented case solution is 1D
Appendix #4
Wall with thicknees
It is possible to create automatically wall thicknesses
Without creating new geometry nor meshing

PLATE

ash wall h=50cm
1. Select B.C as on picture
2. Add new material ash
3. Then set-up in Boundary conditions settings:
   - wall thickness=0.5m
   - wall material ash
4. Run calculations and see results
1. As you can see, there is also an option for heat generation here. However, this is heat generation in the wall!!! Not in the whole body. Type value Q=100 W/m³. Run Calculation and see the results.
Appendix #5
Comparision with 2D analytial solution
Calculate the following case and compare with analytical solution presented on next slide for height $y=0.5m$
For presented case the analytical solution is as follows:

\[
T(x, y) = \frac{2}{\pi} \sum_{n=1}^{\infty} \frac{(-1)^{n+1} + 1}{n} \sin \left( \frac{n\pi x}{L} \right) \frac{\sinh \left( \frac{n\pi y}{L} \right)}{\sinh \left( n\pi \frac{W}{L} \right)}
\]

where

- \(x, y\) are coordinates in X and Y direction
- \(L, W\) plate length and width – here =1m
- \(n\) – integer number
In order to create plot line in the middle of plate:
- go to the XYPlot menu
- select New surface
- choice Line/Rake…
- Set-up line position x0,y0,x1,y1 as bellow

-give any name and click Create and Close window
Now it is possible to select new Surface line 0.5m to plot results:

- Plot results to see Temperature distribution
- Then enable **Write to File** and save data into $T_{2D}.dat$ file for future comparison with analytical solution (presented before)