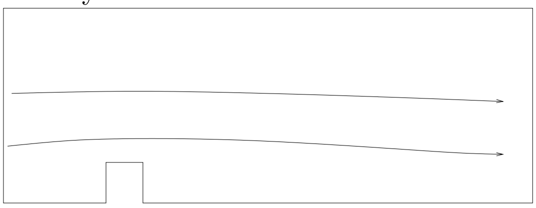
1 Introduction

In this exercise you will calculate the flow around a 2D obstacle (see the figure below), including heat transport. You will make the geometry from scratch, mesh it, and calculate the flow around it, and visualize the results.

You should be able to follow all the steps in this text by yourself. However, feel free to ask questions about the package or the flow you are calculating. At the start, there is lots of comment on what you have to do. Further below, the steps to follow are given only very briefly.

At a certain point of the exercise, we have calculated the flow. The ultimate goal of the CFD package exercises is, to study a flow. You will see examples below. Study the results from a CFD package in any way the package offers (contour plot, vector plot, 1D cuts, total force on an object) and try to find out if the results make sense.



NOTE: all computers can go down. Save your results every now and then.

First we start with gambit to make the geometry. The package gambit allows you to define the geometry, boundary conditions and the grid. Later on we use the package fluent to perform calculations and visualize the results.

Click, in the Start menu:
Programs
Data analysis & Simulation Software
Fluent.Inc
Gambit 2.1.6
Gambit

It can take a minute or two before the program has completed the start-up, you see the gambit window then.

In gambit, click Solver on the main menu, a drop-down list appears, click Fluent 5/6.

2 Saving results

Save your results every now and then. Under the Main menu, click file, and you see the Save and Save As button. Use a filename like C:\TEMP\obstacle.dbs (extension dbs for data base). Please use the C:\TEMP directory or files may be completely lost.

3 Definition of the geometry.

The geometry is built up by first defining points, the points are connected by lines and the lines are used to define a 2D face. This is not always the recommended way to make a geometry, but it will show you some parts of the package you do not already know.

Go to the icons at the upper right. In the lower row (with the word geometry) there are five icons. The left one shows a cube with a fat point. Click on this one.

A row of icons pops up with the title: Vertex. We create eight points needed to define the geometry. Gambit already shows you the window for creating points. We use the method of inserting the coordinates.

For the first point, go to the white spaces below the title: Global. The numbers given are 0, 0 and 0 for x, y and z. This is the first point. Click on the Apply button. In the graphics window, a small white cross appears at the position of the first created point.

Go back to the Create Real Vertex window. Now put (under Global) the value 15 for x and leave the other 0, and click on the Apply button. You may have to click on the white space to the right of the x: before you can change the number. Click on Apply. The point just created may be outside the graphics window. We will re-scale so that all geometry created so far fits within the graphics window. Go to the icons at the lower-right of the main window (with the title: Global Control.) If you do not see them, move the Create Real Vertex window upwards. It shows you a row of small icons with the title Active (they can be used for changing the view angle, we do not use them now). Then there are two rows of larger icons, with the title: Description. Go to the top row of these and hold your mouse button on the left most icon (without clicking). Note, that in the empty white space directly to the left of this icon you get a message from gambit:

FIT TO WINDOW

and some more comment. gambit explains the usage of the icon to you, this works for all icons. (One of the most important one is the right most one of the same row as the FIT TO WINDOW icon: it contains a blue curved arrow. It is the UNDO button, for correcting mistakes.). Go back to the FIT TO WINDOW icon and click, you see both points now.

Create the following points:

x	y	z
0	7	0
15	7	0

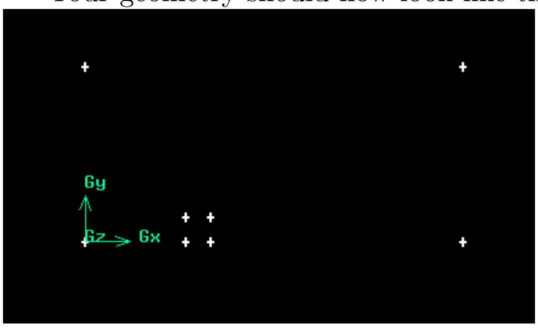
Table 1: Two points for geometry.

Now create points for defining the obstacle:

x	y	z
4	0	0
4	1	0
5	1	0
5	0	0

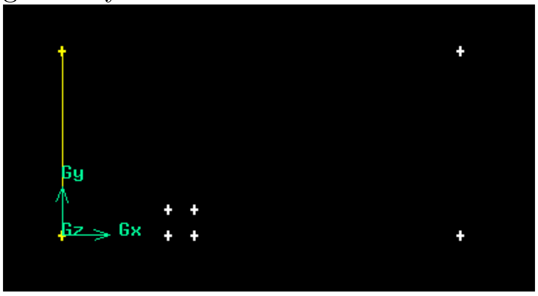
Table 2: Four points defining the obstacle.

Your geometry should now look like this:

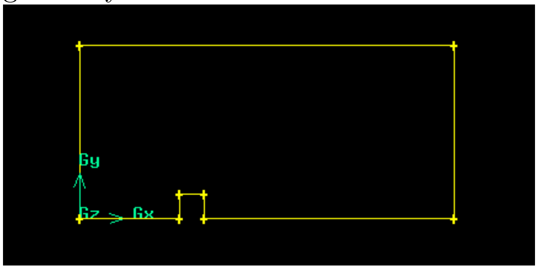


Next we connect the lines with curves. Go to the upper right of the main window. Go to the second row of icons (from the top, with the title: Geometry) and click on the icon with the cube with one thick line. New icons pop up for creating edges, and a window pops up with the title: Create Straight Edge. We will use this window to create the edges by connecting pairs of points.

Go to the yellow space on the Create Straight Edge window, and click it. We can now choose the points we want to connect. Choose the bottom-left corner point by clicking it with your left mouse button while holding down the shift key at the same time. The white cross turns red now, and the name of the point (which is vertex.1 or vertex.2 or something similar) appears in the yellow space of the window. Now go to the point at the upper left of the geometry, and click it too with the left mouse button while holding down the shift key (this is called: shift-left-click). It also turns red now (the other point may get pink at the same time). Click on Apply in the Create Straight Edge window. The two points are now connected by a (yellow) line. Your geometry now looks like this:

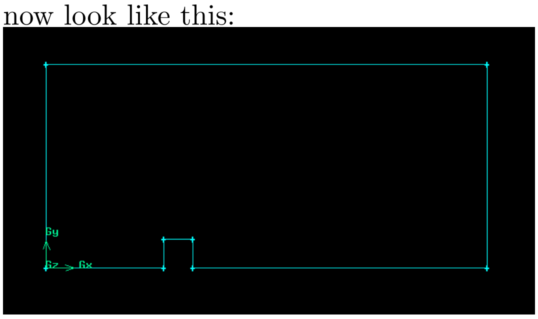


Next create the other edges in a similar way, so that you get the following geometry:



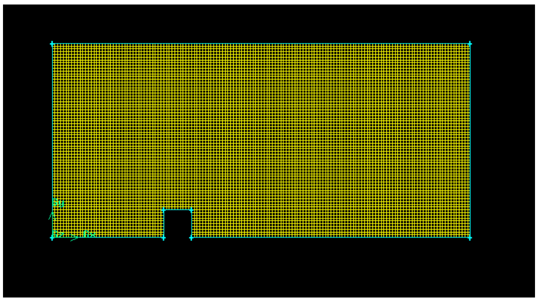
The edges we need have now been created. Next we use the (1D) edges to create a 2D face. Again go to the top-right of the main window and to the second row of icons from the top (with the title: Geometry). Click on the middle icon of this row (it shows a cube with one white face). We now see the two rows of icons for choosing face creation tools (under the title: Face). Now we choose the upper row of the Face creation icons, and click the left one (the Description window shows the text FORM FACE and a short explanation). A window with the title: Create Face from Wireframe appears. (If not, the FORM FACE button was already active, and by clicking it twice we have in fact de-activated it. In this case, activate it again by clicking it once more.)

The Create Face from Wireframe window has a yellow space with the title: Edges. Click it. We can now choose the edges to form a 2D face. One way is clicking them one after the other. The other way is to collect all of them at one (the term is: to grab them). Grabbing is done as follows: go to a point to the left of all the edges, and below them. shift-left-click the mouse there, and keep the shift button and left mouse button pressed. Then move the mouse to a point above all edges, and to the right of all edges. Overthere, release the left mouse button. All edges colour red now. Click the Apply button on the Create Face from Wireframe window. The colour of all the edges now turns blue. We have created a 2D face. The geometry is ready now, and we are going to put a grid on it. The geometry should now look like this:



Go to the icons at the upper right of the main window. From the upper row of icons (with the title: Operation) click on the one with a yellow cube. Another row of icons appears, with the title: Mesh. Click on the middle one (showing a cube with a white face). The Mesh Faces window pops up. Click the yellow space to the right of the title: Faces in this window, and then shift-left-click the face we have created. This turns red now. Go to the white space with the title: Spacing. Put the spacing equal to 0.1 and click the Apply button.

The grid is generated. Use the FIT TO WINDOW icon near the bottom-right of the main window to fit the geometry to the window. The geometry should now look like this:



We have now completed the geometry and the grid, and will turn to the boundary conditions.

4 Definition of the boundary conditions.

As boundary conditions we take prescribed velocity at the left, outflow at the right and a symmetry boundary condition at the top. At a symmetry boundary, we prescribe the normal velocity to 0 and allow a slip velocity for the tangential components so that the wall shear stress is 0. All other parts of the boundary get the wall boundary condition, meaning that the velocity is 0 there.

Go to the top-right of the main window, to the first row of icons. Click the icon with the blue cube, you now see another row of two icons appearing. Click on the left one (with the white cube) of the new row. The Specify Boundary Conditions window pops up.

Click the white space to the right of the title: Name: and put the text inflow there. Click the button with the title Type: on it (which shows the word WALL now). A list drops down, go to the bottom of the list and click on Velocity Inlet. Unfortunately, Velocity Inlet is the lowest item, just under Symmetry, and may not be visible (a bug in gambit). Choose Symmetry, and then click the button Type: again. The drop down list appears again, this time with the item Velocity Inlet visible. Choose Velocity Inlet. Now go to the yellow space with the title Entity: To the left of the yellow space there is a button showing the word Faces. Click the button, a list drops down. Click on edges (since we do a 2D geometry, we specify boundary conditions on edges rather than faces). Now click on the yellow space, so we can choose an edge, and shift-left-click on the leftmost edge. It will turn red. Click on the Apply button on the Specify Boundary Conditions window.

Next go to the white space with the title Name: and put the name outflow there. Go to the Type: button and change VELOCITY_INLET to

OUTFLOW. Go to the yellow space with the title Edges: and click it, then shift-left-click the rightmost edge. It turns red. Now click the Apply button on the Specify Boundary Conditions window.

Now create the symmetry boundary, with name symmetry_top, with type: SYMMETRY and the topmost edge as the location (do not forget to click the Apply button at the end).

Now we want to create wall boundaries. If you specify nothing in gambit, a boundary edge is automatically of type WALL. We will still explicitly create wall boundaries, because we want to be able to distinguish several parts of the wall boundary as a whole.

First we create a wall boundary condition on the obstacle. Give it Name: wall_obstacle, give it type: wall and click all three edges forming the obstacle, one after the other. Then click Apply.

Next create a wall boundary condition (name: wall_before_obstacle) at the bottom edge in front of the obstacle and create a wall boundary condition (name: wall_after_obstacle) at the bottom edge after the obstacle.

5 Writing the grid and saving the session

We are now ready with the geometry, and will write the grid and save the session. Click on File at the top menu bar, and select Export on the drop-down list. Another drop-down list appears, select Mesh. A window pops up, type in a name (for instance, C:\TEMP\obstacle.msh) VERY IMPORTANT: click "Export 2D mesh", otherwise nothing is written. Then click OK. In the white space to the bottom-left (title: Transcript) now shows the comment: Mesh successfully written. Return to the main menu bar, click File, then Save As on the drop-down list which appears. The Save Session AS window pops up, type a name (for instance: C:\TEMP\obstacle.dbs) and click on the Accept button. Again, go to the main menu bar, click File and choose exit. The Exit window pops up, asking you if you want to save the session. You just did, click the No button.

6 Running fluent.

We now continue with fluent to calculate the flow. Start fluent as follows:
Under Start:
Programs
Data analysis & Simulation Software

Fluent.Inc
fluent 6.1.22
fluent 6.1.22
A fluent window pops up. It contains a white space with text and a prompt with the title: versions. It already selects 2d and this is what we want, so click run. The main fluent window pops up. Now go to the menu bar at the top of the window. Click file. A list drops down, click on read. Yet another list drops down. Click on case. A window pops up, showing the working directory and some file names. Go to C:\TEMP Click the name: obstacle.msh (or any other name you have chosen) and this name will now selected (together with the whole directory path) in the appropriate white space. Click OK. Fluent reacts by reading in the mesh file listing information about the grid and the boundary conditions.

7 Saving results

This is done under the main menu: file, write case & data. If you have no data yet, file, write, case. Choose suitable file names, make sure the files are under C:\TEMP Adding extension .gz will make fluent automatically zip the files.

8 Checking the mesh.

It is good practice to check on the grid, especially for complicated geometries. Go to the menu bar at the top of the window and click in grid A list drops down, click on check. Fluent reacts by giving general grid info. One of the things is says is the minimum and maximum value. You may need to use the scroll bar to see these values. In case the minimum value is negative, the grid contains errors.

Reorder the points. Fluent uses a numbering of the grid points which can sometimes be optimized. Click: grid, then on the pop-down list click: reorder. A small list appears, click on domain. Fluent reacts by giving some info on the reordering operation.

9 Setting the working conditions.

We have to tell the package in what flow regime (laminar, turbulent) we work. Go to the menu bar at the top of the window, and click on define.

On the pop-down list, click on models. Another list pops down. Click on viscous. A window pops down, showing the names of various turbulence models and the options inviscid and laminar. Click on: laminar. Then OK. We include the temperature in the simulation. Do: Define-Models-Energy. A window pops up to ask you if you want to include the temperature in the simulation. Choose Energy Equation. Click OK. Click on OK.

10 Adapting the fluid properties.

We want a low Re number flow. To get a laminar flow we take a bigger viscosity. Modify the viscosity of the working fluid (which is air by default) as follows. Click define (on main menu bar), then materials. A menu with properties of air pops up. Set the viscosity to 0.1. Click Change/Create, Close.

11 Setting the boundary conditions.

Go to the menu bar at the top, click define and then boundary conditions. A window pops up showing the words default-interior, fluid, inflow, outflow, symmetry_top, and wall_obstacle, wall_before_obstacle and wall_after_obstacle. Default-interior has been set by fluent and is not relevant now. If you click on fluid and then click on set, a window appears in which you can set for instance gravitation (not used now). Next click on inflow and then on set (at the bottom of the window). We have to give information about the inflow speed and the temperature at the inlet. Click on the white space to the right of: Velocity Magnitude, and change the 0 to 1 (giving 1 m/s). The temperature is 300K at the inlet, set this too. Now click the OK button. You are now back to the boundary conditions window. Choose wall_obstacle, choose set. The boundary condition for temperature is changed from zero flux to temperature. Choose temperature 250. Click OK. You are now back to the boundary conditions window. We are finished with it, so click on the close button.

12 Setting and running the solver.

We are now going to set some info for the solver. Go to the top menu bar. Click on solve. Then click on initialize. A drop-down list appears. Click on: initialize in this drop down list. A window pops up in which you can initialize the solution. The closer the initial values are to the real solution, the faster the solver will converge. We will let fluent calculate the initial condition from the inlet values. Click on the arrow near the Compute From button, and choose inflow. The values at the white spaces now change to the values given at the inlet. Click on the init button and on the close button at the bottom of the window.

13 Setting solution monitors.

For some quantities we want to monitor how they change during the calculation. For instance, we want to know the force exerted on the obstacle by the fluid. Fluent can calculate this for you and can check the evolution of the force during the solution. For instance, this can be a check if the solution is accurate enough.

Go to the main menu, click on Solve, then on Monitors on the drop-down list which appears. The another drop-down list appears, click on Force. The Force Monitors window pops up. Among other things, there is a white space with the title Wall Zones, showing a list of all entities which are declared WALL. Near the middle left there is a button called Coefficient. It says Drag. Leave it that way. Near the bottom of the window there is a white space with the title File Name. Leave it to cd-history. In the upper-left part of the window there are buttons for choosing the kind of output. Click on Plot. Click on wall_obstacle which is then marked blue. Then click the apply button. We also want to have a look at the lift. Click on the arrow to the right of the Coefficient button (which has the name Drag now). A drop down list appears, change Drag to Lift. Click on wall_obstacle (the blue mark has disappeared when we changed the Coefficient from Drag to Lift). The File Name is cl-history, leave it that way. Choose the Plot button at the left to get screen output. Then click apply. Close the Force Monitors window. We can scale the output of the force coefficients. Go to the main menu bar, click on Report, and click on Reference Values on the drop-down menu which appears. Normally, the force coefficient of an obstacle is scaled like:

$F_{scaled} = F/\rho A U^2$ where F is the force, A the front area and U a typical velocity and ρ the density. The front area is 1 in out case, a typical velocity (the free stream velocity) is 1, and ρ is the density of air. All values shown are correct for this problem, so just click OK.

We also monitor the heat flux through the surface of the obstacle.

Solve-Monitors-Surface
set Surface Monitors to 1 and do enter
click Plot for monitor-1
click define
select (under Surfaces) wall_obstacle
Report Type: Area Weighted Average
Plot Window 2
select Wall Fluxes under Report of
under this last line, select Total Surface Heat Flux
OK in define window
OK in surface monitor window.

And, finally, the residuals are monitored. The discrete equations are solved iteratively. The residual gives you an idea how far the solution is converged. Smaller residual means closer to convergence. Solve-Monitors-Residual
select Plot
Plot Window 3
OK

NOTE: during the run windows will pop up in which the monitor quantities are plotted. One window my be on top of the other, so you may have to shift them to see them all.

14 Running the calculation.

We are now ready to start a calculation. Go to the top menu bar and click on solve again. A pop-down list appears. Click on iterate. A window now appears. Beside the title: Number of iterations, click on the white space. Change the number of iterations to 100. Now click on the Iterate button at the bottom of the window to start the solver. Fluent reacts by starting the solver. At the same time, three extra windows pop up which show the drag and lift on the obstacle and the average heat flux. Note that after some initial variation they start to converge to a steady value. In the fluent window you see numbers appearing which give a measure of the error. Numbers appear for continuity (=mass conservation), x-velocity and y-velocity and energy (temperature). Depending on the physical parameters of the problem, after 100 iterations the solution may not yet be converged. If so, click on the Iterate in the Iterate window again (and more times if necessary). Every time you click on Iterate again, fluent does another 100 iterations, starting from the last result. After several times clicking the iterate button the solution is converged. However, you see that the force on the obstacle is still varying. Moreover, look at the scale of the variation of the force. The force is changing very rapidly at the beginning, so it is not easy to see any variation during the later stages. If you have time, you can improve on this. If you do not have enough time left (30 minutes or less), skip the following part and continue at the text RESUME HERE.

15 If you have time: clear the monitors and rerun with smaller residual.

We reset the monitor windows, so that we can see small changes, and give a more severe convergence criterion.

To reset the monitor windows:
solve
monitors
force
click clear, and YES on the pop-up window
change drag to lift (or vice versa, depending on what you did first)
clear, OK

The calculation stops when it has reached a certain accuracy. For some problems this may not be accurate enough. You have to ask Fluent to get smaller residuals then. Solve, monitors, residual
Change the number 0.001 appearing as convergence criterion for u, v and continuity to the smaller value 0.00001 (1e-5)
OK

Restart the solver (it will use the old results)
Main menu:
Solve
Iterate
Iterate

Give some more iterations if you want better convergence, if you are satisfied stop.

Click on the close button in the iteration window.

RESUME HERE.

16 Visualizing the results.

Go to the top menu bar and click on display. A drop-down menu appears. First we have a look at the geometry itself. Click on grid. A window appears in which you can choose what you want to see. Just click on display. A black window pops up showing the geometry. Click on display (top menu bar) again. Click on contours on the drop-down menu. You now get a window (the Contours window) in which you can choose what variables (pressure, velocity) you want to visualize, and on which parts of the geometry (for instance, inflow boundary, default-interior (= region where the fluid flows).

Click on the Display button at the bottom of the window. You see contour lines of the pressure. If you prefer a picture with filled contours instead of contour lines, go to the Contours window. To the left, near the top there is a button with the title Filled. Click it. Next click on Display.

Study the pressure plot. Where is the pressure big, where small? Is this as you expect it to be? Go back to the Contours window (where you clicked the Display button). At the upper right you see several button to choose the variable which you want to visualize. Click the arrow to the right of the title Pressure... On the drop down menu click Velocity. The title of the button changes to Velocity, and the title of the button below it changes too. On the other button you now have the title Velocity magnitude. Click in the arrow to the right of the title Velocity magnitude. A long drop-down menu appears, showing all kinds of quantities related to velocity. Click on x-velocity. Next click on Display. We see a contour plot for the x-velocity now.

16.1 Analysis of the results

Note several things in particular: where is the velocity greatest, where is it smallest? From the legend you can see that there are large parts of the flow where the velocity is negative. This is especially so to the right of the obstacle, where there is a large wake. To see the extent of the region of negative velocity, go to the Contours window. Click on the Auto Range button (which is then de-selected). The button below it (with the title Clip to Range) is now automatically selected. To the right of the Auto Range and Clip to Range buttons are two white spaces, with the titles Min and Max. They give the minimum and maximum value of the velocity in the contour plot. Change the maximum to 0 so that only the negative part is shown. Click Display. As you can see, the negative region extends almost up to the exit. You can zoom in to a region of interest (for instance, the top of the cube) by putting the mouse under the region of interest, and to the left of it. Click the middle mouse button and keep it pushed in. Move to a point above, and to the right of the region of interest. You will see that fluent shows a rectangle going from the point that you cntrl-middle-clicked first to the current mouse position. Release the mouse button and the control button if the rectangle covers the region of interest. The visualizer now zooms in to the rectangle you indicated. You can go back to the original view (of the whole geometry) by clicking on Display on the main menu bar, click Views on the drop-down list and then the Views window pops up. Click the Default button under Actions. Next click the Close button to close the window.

16.2 Flow pattern analysis

In 2D you can use the stream function to display stream lines.
Main menu: Display-Contours, choose Velocity, type of velocity plot: Stream function, Display on the Contours window.

You see the stream lines. They do not show up behind the obstacle. You can release particles to see what is happening there. First you must tell the package where to release the particles. Define a line segment for that.

Main menu: surface-LineRake. You get the LineRake Surface Menu. A line is defined by two points, which then cuts through the whole fluid. A rake is a line segment, which only exists between the two points you give.

Create a line and a rake. First a line. Type Line is default. Choose points (x0,y0) = (4.5,0) and (x1,y1) = (4.5,1) which is a vertical line through the obstacle. Note the name of the line in the New Surface Name space. The z coordinate is disabled (we work 2D). The rake: change Type Line to Type Rake, take points (x0,y0) = (6,0) and (x1,y1) = (6,1). Note the name of the rake in the New Surface Name space.

Check line and rake. Main menu: Display-Grid. De-select default-interior, only the outline of the geometry is then shown, not the grid. Display. Now also select the rake and the line. Display. You see them now in the picture.

Display-Path Lines. Select the line in the Release From Surface space. Note, that Step Size and Steps (number of steps) have pre-defined values, set Step Size to 0.1. Display.

De-select the line (by clicking the line agian) and now select the rake in the Release From Surface space. Display. Note the re-circulation region.

16.3 A vector plot

As a last visualization aid, we have a look at a vector plot. Go to the main menu bar, and click on Display, and click on Vectors on the drop-down list which appears. The velocity vectors window pops up, at the right-middle of this window there is a white space with the title Surfaces. It contains a list of geometric entities which we can select to draw velocity vectors. We want to see them trough the whole fluid domain, so click on default-interior. Then click on the Display button.

The number of vectors is so large that it may be hard to see separate vectors. Zoom in to a region near the top of the obstacle. Again, you can see the re-circulation region overthere. You can go back to the default view (Display-Views-Default) and have a look at other regions of interest, like the outflow, or other sides of the obstacle.

17 1D plots and numerical data.

So far, we have made overview pictures which give a general idea of the flow. We also want to make comparisons with experiments or get numerical data (real numbers). We do this for 1D plots, which are easiest to visualize and interpret.

Close all window except the fluent window and the black window showing the geometry.

We want to have a look at data along a vertical line through the mid of the geometry. Plot-XY Plot
select the line you made earlier
plot variable: Temperature-Static Temperature
Plot
Your plot is not meaningful: first set, under Plot Direction:
X to 0
Y to 1
Plot
You see the temperature profile on top of the obstacle. Observe the very sharp gradient near the obstacle. You get a less steep one by adapting the value of the thermal conductivity and running the solver. Under Main Menu: define

materials
make Thermal Conductivity 10 times as large
re-run the solver:
Iterate
Iterate
Convergence is very quick! Only the temperature changes. Plot-XY Plot curve and plotting variable (temperature) are still selected, so:
Plot
The gradient near the top of the obstacle is smoother now. Repeat for a conductivity which is again 10 times as large, and see what happens. Can you explain what is happening?

If we want to save the data in numerical form, go to the upper-left part of the window and click the Write to File button. The Plot button at the bottom of the window now turns into a Write button Click this Write button. A window appears with a white space (title XY file) in which we have to type a file name. Click the white space and type (for instance) velocity.dat Next click the OK button at the bottom of this window.

18 Ending the session: Writing the data and leaving fluent.

Click on the close buttons of all the windows, until only the main fluent window is open. On this window, click the File button in the main menu bar. Click on Write, a pop-down list appears. Click on Case & Data. A window pops up which has a white space (with the title Case/Data File) in which you can type a file name (say, C:\obstacle.cas). To save memory, you can type C:\obstacle.cas.gz with extra extension .gz then fluent will zip the files automatically on writing. Click the OK button. Fluent reacts by giving info on the data written. Now click on File in the main menu button, and the exit. Fluent now exits and closes any windows which are still open.

If you want to keep the results, copy or move them to your own disk!