SMARTFIRE v4.3

USER GUIDE AND TECHNICAL MANUAL
SMARTFIRE TUTORIALS

BY

J. EWER, F. JIA, A. GRANDISON,
E. GALEA and M. PATEL

September 2013
Document revision 4.3.1

TABLE OF CONTENTS

25 *SMARTFIRE* TUTORIALS 25-1

25.1 INTRODUCTION TO TUTORIALS 25-1

25.2 TUTORIAL 1 25-2
25.2.1 OVERVIEW 25-2
25.2.2 STEP 1: LOADING AN EXISTING GEOMETRY SPECIFICATION 25-2
25.2.3 STEP 2: CREATING A MESH FOR THE SIMULATION 25-4
25.2.4 STEP 3: RUNNING THE CFD ENGINE 25-7
25.2.5 STEP 4: EXITING THE CFD ENGINE 25-11

25.3 TUTORIAL 2 25-13
25.3.1 OVERVIEW 25-13
25.3.2 STEP 1: STARTING THE CASE SPECIFICATION TOOL 25-13
25.3.3 STEP 2: CREATING THE DOMAIN 25-14
25.3.4 STEP 3: CREATING GEOMETRY OBJECTS 25-14
25.3.5 STEP 4: DEFINING THE PROBLEM TYPE 25-21
25.3.6 STEP 5: SAVING TO A NEW CASE DIRECTORY 25-22

25.4 TUTORIAL 3 25-24
25.4.1 OVERVIEW 25-24
25.4.2 STEP 1: STARTING THE CASE SPECIFICATION TOOL 25-24
25.4.3 STEP 2: CREATING THE DOMAIN 25-25
25.4.4 STEP 3: CREATING GEOMETRY OBJECTS 25-25
25.4.5 STEP 4: DEFINING THE PROBLEM TYPE 25-32
25.4.6 STEP 5: SAVING SPECIFICATION TO A NEW CASE DIRECTORY 25-33
25.4.7 STEP 6: FINISH SPECIFICATION AND START CFD ENGINE 25-34

25.5 TUTORIAL 4 25-36
25.5.1 OVERVIEW 25-36
25.5.2 STEP 1: CREATING THE GEOMETRY AND SCENARIO 25-37
25.5.3 STEP 2: CREATING A MESH FOR THE SIMULATION 25-45
25.5.4 STEP 3: RUNNING THE CFD ENGINE 25-49
25.5.5 STEP 4: INTERPRETING RESULTS FROM SMARTFIRE 25-55
25.5.6 STEP 5: EXITING THE CFD ENGINE 25-56

25.6 TUTORIAL 5 25-57
25.6.1 OVERVIEW 25-57
25.6.2 STEP 1: LOADING THE BASE CASE AND RENAMING 25-58
25.6.3 STEP 2: ADDING A DUCTED FAN TO THE SCENARIO 25-59
25.6.4 STEP 3: CREATING A MESH FOR THE SIMULATION 25-63
25.6.5 STEP 4: RUNING THE CFD ENGINE 25-67
25.6.6 STEP 5: INTERPRETING RESULTS FROM SMARTFIRE 25-74
25.6.7 STEP 6: EXITING THE CFD ENGINE 25-75

25.7 TUTORIAL 6 25-76
25.7.1 OVERVIEW 25-76
25.7.2 STEP 1: LOADING THE BASE CASE AND RENAMING 25-78

25.7.3 STEP 2: ACTIVATING THE COMBUSTION MODEL 25-78
25.7.4 STEP 3: ACTIVATING THE MULTIPLE RAY RADIATION MODEL ... 25-81
25.7.5 STEP 4: CREATING A MESH FOR THE SIMULATION 25-82
25.7.6 STEP 5: RUNNING THE CFD ENGINE 25-85
25.7.7 STEP 6: INTERPRETING RESULTS FROM SMARTFIRE 25-91
25.7.8 STEP 7: EXITING THE CFD ENGINE 25-92

26 *SMARTFIRE* ADVANCED TUTORIALS 94

26.1 INTRODUCTION 94

26.2 TUTORIAL A1 95
26.2.1 GEOMETRY SPECIFICATION 95
26.2.2 SCENARIO SPECIFICATION 95
26.2.3 MONITORING 96
26.2.4 DATA GATHERING 97
26.2.5 MESH AND SIMULATE 97
26.2.6 RESULTS AND ANALYSIS 98

26.3 TUTORIAL A2 99
26.3.1 RUTIME DATA ANALYSIS IN THE INTERACTIVE CFD ENGINE 99
26.3.2 DATA EXPLORER 99
26.3.3 PLOT GRAPHS 100
26.3.4 FUNCTION SOLVER 101

26.4 TUTORIAL A3 105
26.4.1 POST PROCESSING VISUALIZATION USING DATAVIEW 105
26.4.2 LOAD THE GEOMETRY 105
26.4.3 PREPARE FOR ANIMATION 106
26.4.4 CREATE AN ANIMATED SCALAR VISUALIZATION 106
26.4.5 CREATE AN ANIMATED VECTOR VISUALIZATION 107
26.4.6 VISUALIZATION NOTES 107

26.5 TUTORIAL A4 108
26.5.1 COMBUSTION MODELLING 108
26.5.2 MODIFY TUTORIAL A1 SCENARIO 108
26.5.3 CASE SPECIFICATION 109
26.5.4 MESH AND RUN THE SIMULATION 110
26.5.5 RESULTS ANALYSIS 110

26.6 TUTORIAL A5 112
26.6.1 CAD IMPORT TUTORIAL 112
26.6.2 CREATE FLOOR PLAN 113

26.7 TUTORIAL A6 115
26.7.1 USING PARALLEL SMARTFIRE 115
26.7.2 LOAD DXF FLOOR PLAN 115
26.7.3 CREATE SCENARIO 116
26.7.4 RUN THE PARALLEL CFD SIMULATION 117
26.7.5 EXERCISES FOR PARALLEL SMARTFIRE TUTORIAL 118

TABLE OF FIGURES

FIGURE 25-1 SPECIFICATION ENVIRONMENT SHOWING THE "A74_CASE" GEOMETRY. 25-3
FIGURE 25-2 – GEOMETRY TYPE MESHING RULES SELECTION DIALOG................................ 25-4
FIGURE 25-3 ADDITIONAL EXTENDED REGION DIALOG 25-5
FIGURE 25-4 MESHING TOOL SHOWING SELECTION OF RECOMMENDED CELL BUDGET......... 25-6
FIGURE 25-5 MESHING TOOL SHOWING A FOUR VIEWS OF THE MESH AND GEOMETRY. 25-7
FIGURE 25-6 *SMARTFIRE* CFD ENGINE USER INTERFACE AT START UP. 25-8
FIGURE 25-7 VISUAL CONFIGURATION SHOWING SLICE AND DISPLAY OPTIONS................. 25-10
FIGURE 25-8 VISUALISATION OF TEMPERATURE AND VELOCITIES FOR MID Z-SLICE. 25-11
FIGURE 25-9 SPECIFICATION TOOL SHOWING AN EMPTY DEFAULT INITIAL REGION............ 25-13
FIGURE 25-10 SPECIFICATION TOOL SHOWING ENTRY OF CORRECT REGION SIZES. 25-14
FIGURE 25-11 SPECIFICATION TOOL SHOWING OBJECT EDITOR PANEL. 25-15
FIGURE 25-12 NEW OBJECT SELECTION DIALOGUE USED TO CREATE A NEW VENT........... 25-15
FIGURE 25-13 SPECIFICATION TOOL SHOWING NEW VENT AT DEFAULT LOCATION. 25-16
FIGURE 25-14 SPECIFICATION TOOL USING THE "UNFOLDED" PLANE SELECTOR................ 25-17
FIGURE 25-15 SPECIFICATION TOOL SHOWING CORRECT VENT SIZE AND POSITION........... 25-18
FIGURE 25-16 NEW OBJECT DIALOGUE USED TO CREATE A NEW SIMPLE_FIRE. 25-18
FIGURE 25-17 SPECIFICATION TOOL SHOWING DEFAULT SIMPLE_FIRE.......................... 25-19
FIGURE 25-18 SPECIFICATION TOOL SHOWING FIRE POSITION AND SIZE. 25-20
FIGURE 25-19 FIRE PROPERTIES DIALOGUE SHOWING DEFAULT FIRE SETTINGS. 25-20
FIGURE 25-20 FIRE PROPERTIES SHOWING A LINEAR FIRE CURVE TO 62.9KW IN 20.0S. 25-21
FIGURE 25-21 EXPERT PROBLEM TYPE SHOWING THE TYPE OF SIMULATION. 25-22
FIGURE 25-22 USING FILE "SAVE AS..." TO SPECIFY A NEW CASE NAME. 25-23
FIGURE 25-23 SPECIFICATION TOOL SHOWING DEFAULT SIZED REGION. 25-24
FIGURE 25-24 SPECIFICATION TOOL SHOWING CORRECTLY SIZED REGION......................... 25-25
FIGURE 25-25 SPECIFICATION TOOL SHOWING THE OBJECT PANEL..................................... 25-26
FIGURE 25-26 SPECIFICATION TOOL SHOWING NEW VENT AT THE DEFAULT LOCATION. ... 25-27
FIGURE 25-27 SPECIFICATION TOOL SHOWING DOOR POSITION AND SIZE........................... 25-28
FIGURE 25-28 SPECIFICATION TOOL SHOWING WINDOW POSITION AND SIZE...................... 25-29
FIGURE 25-29 SPECIFICATION TOOL SHOWING THE PARTITION. 25-30
FIGURE 25-30 SPECIFICATION TOOL SHOWING A PORTAL CONNECTING THE TWO COMPARTMENTS THROUGH THE PARTITION.. 25-30
FIGURE 25-31 SPECIFICATION TOOL SHOWING FIRE LOCATION AND SIZE. 25-31
FIGURE 25-32 FIRE PROPERTIES WINDOW SHOWING CONSTANT FIRE OF 100.0KW. 25-32
FIGURE 25-33 PROBLEM TYPE DIALOGUE. 25-33
FIGURE 25-34 FILE [SAVE AS] DIALOGUE BEING USED TO GIVE A NAME TO THE CASE........ 25-34
FIGURE 25-35 CFD ENGINE AT END OF FULL SIMULATION (REARRANGED WINDOWS). 25-35
FIGURE 25-36 PLAN VIEW DRAWING SHOWING THE ROOM LAYOUT GEOMETRY................. 25-36
FIGURE 25-37 PROBLEM TYPE OPTIONS WINDOW.. 25-38
FIGURE 25-38 OBSTACLE PROPERTIES USED IN THE SIMULATION SCENARIO. 25-40
FIGURE 25-39 SIMPLE FIRE PROPERTIES WINDOW SHOWING THE FIRE SETTINGS................ 25-40
FIGURE 25-40 MONITOR LINE PROPERTIES WINDOW SHOWING THE SETTINGS. 25-41
FIGURE 25-41 RENDERED PERSPECTIVE 3D GEOMETRY SHOWING WINDOWS, WALLS, FIRE AND MONITOR LINE.. 25-42
FIGURE 25-42 GEOMETRY SHOWN IN WIRE FRAME VIEW MODE. 25-42
FIGURE 25-43 PLAN VIEW (XZ) DISPLAY OF THE GEOMETRY.. 25-43

FIGURE 25-44 SIDE VIEW (XY) DISPLAY SHOWING THE GEOMETRY.................................. 25-43
FIGURE 25-45 END VIEW (YZ) DISPLAY SHOWING THE GEOMETRY................................... 25-44
FIGURE 25-46 MESHING CONTROL PARAMETERS WINDOW. 25-45
FIGURE 25-47 WINDOW SHOWING THE OPTIONS FOR CREATING EXTENDED REGIONS. 25-46
FIGURE 25-48 CELL BUDGET FOR MESHING. 25-47
FIGURE 25-49 FIRST ATTEMPT COURSE MESH FROM THE AUTOMATED MESHING SYSTEM. .. 25-48
FIGURE 25-50 REFINED MESH. 25-49
FIGURE 25-51 *SMARTFIRE* CFD ENGINE USER INTERFACE AT START UP. 25-50
FIGURE 25-52 VISUAL CONFIGURATION SELECTING THE LOWEST Y-LAYER OF CELLS TO VIEW. 25-51
FIGURE 25-53 VISUALISATION OF THE LOWEST Y-SLICE OF CELLS SHOWING THE FIRE LOCATION. 25-51
FIGURE 25-54 *SMARTFIRE* USER INTERFACE AFTER SELECTION OF A MORE INTERESTING VISUAL SLICE PLANE THROUGH THE FIRE. 25-52
FIGURE 25-55 DATA CAPTURE CONFIGURATION MENU. 25-53
FIGURE 25-56 *SMARTFIRE* USER INTERFACE SHOWING THE END STAGE OF THE SIMULATION. 25-54
FIGURE 25-57 GRAPH OF MONITORED TEMPERATURES IN LARGE ROOM AT T=118S........... 25-55
TABLE 25-58 TABLE OF TIMES TO DETECT CRITICAL TEMPERATURES IN MONITORED ROOMS. 25-56
FIGURE 25-59 PLAN VIEW DRAWING SHOWING THE ROOM LAYOUT GEOMETRY.................. 25-57
FIGURE 25-60 SPECIFICATION TOOL SHOWING THE RENAMED CASE FROM TUTORIAL 4...... 25-59
FIGURE 25-61 COMPONENTS USED FOR THE CONSTRUCTION OF A DUCTED FAN USING A SIMPLE FAN OBJECT, AN OUTLET OBJECT AND FOUR THIN PLATE OBJECTS. 25-60
FIGURE 25-62 SIMPLE FAN PROPERTIES WINDOW. 25-61
FIGURE 25-63 CLOSE UP WIRE FRAME VIEW OF THE FAN UNIT. 25-62
FIGURE 25-64 EXPLODED VIEW (WITH COMPONENTS SEPARATED) OF THE FAN UNIT. NOTE THAT THERE IS AN AIR GAP NEEDED BETWEEN THE SIMPLE FAN OBJECT AND THE OUTLET OBJECT. 25-62
FIGURE 25-65 WINDOW SHOWING THE OPTIONS FOR CREATING EXTENDED REGIONS. 25-64
FIGURE 25-66 CELL BUDGET FOR MESHING. 25-65
FIGURE 25-67 MESH FROM THE AUTOMATED MESHING SYSTEM. 25-66
FIGURE 25-68 *SMARTFIRE* CFD ENGINE USER INTERFACE AT START UP. 25-68
FIGURE 25-69 VISUAL CONFIGURATION SELECTING THE LOWEST Y-LAYER OF CELLS TO VIEW. 25-69
FIGURE 25-70 VISUALISATION OF THE LOWEST Y-SLICE OF CELLS SHOWING THE FIRE LOCATION. 25-70
FIGURE 25-71 *SMARTFIRE* SHOWING VISUALS OF THE FIRE PLANE AND THE FAN PLANE... 25-71
FIGURE 25-72 DATA CAPTURE CONFIGURATION MENU. 25-72
FIGURE 25-73 *SMARTFIRE* USER INTERFACE SHOWING THE END STAGE OF THE SIMULATION, SHOWING THE FIRE PLANE VISUAL AND THE MONITORED TEMPERATURE PLOT GRAPH. 25-73
TABLE 25-74 TABLE OF TIMES TO DETECT CRITICAL TEMPERATURES IN MONITORED ROOMS. 25-74
FIGURE 25-75 PLAN VIEW DRAWING SHOWING THE ROOM LAYOUT GEOMETRY.................. 25-76
FIGURE 25-76 SPECIFICATION TOOL SHOWING RENAMED CASE FROM TUTORIAL 4............ 25-78
FIGURE 25-77 PROBLEM TYPE OPTIONS SHOWING THE ACTIVATION OF THE COMBUSTION MODEL. 25-79
FIGURE 25-78 COMBUSTION MODEL OPTIONS. 25-80
FIGURE 25-79 FIRE PROPERTIES FOR THE EQUIVALENT FUEL RELEASE RATE...................... 25-81

FIGURE 25-80 RADIATION MODEL OPTIONS WINDOW SHOWING THE SELECTION OF MULTIPLE RAY RADIATION USING 24 RAYS. 25-82
FIGURE 25-81 CELL BUDGET FOR MESHING. 25-83
FIGURE 25-82 FIRST COURSE MESH FROM THE AUTOMATED MESHING SYSTEM. 25-84
FIGURE 25-83 MANUALLY EDITED AND REFINED MESH. 25-85
FIGURE 25-84 *SMARTFIRE* CFD ENGINE USER INTERFACE AT START UP. 25-86
FIGURE 25-85 VISUAL CONFIGURATION SELECTING THE LOWEST Y-LAYER OF CELLS TO VIEW. 25-87
FIGURE 25-86 VISUALISATION OF THE LOWEST Y-SLICE OF CELLS SHOWING THE FIRE LOCATION. 25-87
FIGURE 25-87 *SMARTFIRE* USER INTERFACE AFTER SELECTION OF A MORE INTERESTING VISUAL SLICE PLANE THROUGH THE FIRE. 25-88
FIGURE 25-88 DATA CAPTURE CONFIGURATION MENU. 25-89
FIGURE 25-89 *SMARTFIRE* USER INTERFACE SHOWING THE END STAGE OF THE SIMULATION. 25-90
TABLE 25-90 TABLE OF TIMES TO DETECT CRITICAL TEMPERATURES IN MONITORED ROOMS FROM TUTORIAL 4 (USING VOLUMETRIC HEAT RELEASE AND SIX FLUX RADIATION). .. 25-91
TABLE 25-91 TABLE OF TIMES TO DETECT CRITICAL TEMPERATURES IN MONITORED ROOMS FROM TUTORIAL 6 (USING COMBUSTION AND MULTIPLE RAY RADIATION USING 24 RAYS). 25-92
FIGURE 26-1 : THREE ROOM GEOMETRY WITH TALL FIRE AND OPEN DOORS. 95
FIGURE 26-2 : THE SMARTFIRE DATA EXPLORER WINDOW. 99
FIGURE 26-3 : THE SMARTFIRE PLOT DEFINITION WINDOW. 100
FIGURE 26-4 : THE SMARTFIRE FUNCTION SOLVER WINDOW. 101
FIGURE 26-5 : USING THE FUNCTION SOLVER WINDOW TO SELECT A SUB-REGION. 102
FIGURE 26-6 : USING THE FUNCTION SOLVER TO SELECT THE CELLS IN A DOORWAY. 103
FIGURE 26-7 : USING THE FUNCTION SOLVER TO CALCULATE THE NET FLUX THROUGH A DOORWAY. 104
FIGURE 26-8 : THE SMARTFIRE DATAVIEW PROGRAM AT START UP. 105
FIGURE 26-9 : DATAVIEW SHOWING LOADED GEOMETRY AND ANIMATION LIST OF DATA SETS. 106
FIGURE 26-10 : DATAVIEW SHOWING TEMPERATURE ISO-SURFACE, TITLE AND TIME. 107
FIGURE 26-11 : TUTORIAL A4 GEOMETRY SHOWING MODIFIED FIRES, DOOR AND TRIGGER CELL. 108
FIGURE 26-12 : SMARTFIRE SCENARIO DESIGNER SHOWING LOADED CAD FLOOR PLAN. .. 112
FIGURE 26-13 : SCENARIO DESIGNER SHOWING ZOOMED SUB-REGION FOR SCENARIO MODEL. 114
FIGURE 26-14 : SIMPLE CAD FLOOR PLAN REPRESENTING TUTORIAL A1 SCENARIO. 116
FIGURE 26-15 : SCENARIO DESIGNER FLOOR PLAN WITH DOOR AND FIRE OBJECTS. 117

25 *SMARTFIRE* TUTORIALS

25.1 *INTRODUCTION TO TUTORIALS*

A number of ***SMARTFIRE*** tutorials have been developed. These are intended to provide users with practice and guidance in the basic aspects of loading a previously saved case, specification of a new case and simulation of fire modelling cases within the ***SMARTFIRE*** CFD engine.

The first tutorial indicates how an existing geometry specification can be loaded into the set-up environment, meshed by the automated meshing system and finally simulated within the ***SMARTFIRE*** CFD engine.

The second tutorial creates the same case as used in the first tutorial but specifies it without loading a previously saved geometry. It also demonstrates how to turn on the six-flux radiation model and how to modify the fire output (power) curve.

The third tutorial demonstrates how to construct multiple compartments within a region (separated by a compound partition) with a fire in one of the compartments.

The fourth tutorial extends the experiences gained by performing the first three tutorials and creates a multiple room geometry with a fire in a corridor. The scenario is used to analyse the times before some of the monitored rooms exceed a critical temperature that is deemed to be hazardous.

The fifth tutorial extends the previous tutorial case to include an extractor fan so that the effect of the extraction of heat can be seen when compared to the previous tutorial.

The sixth tutorial uses the same scenario as for tutorial 4 but uses the gaseous combustion model and the multiple ray radiation model to compare the differences in the solutions when different modelling approaches are adopted.

A set of advanced tutorials is also available. The advanced tutorials provide less detailed assistance than the tutorials in this document, but instead provide the user with example problems with which to practice using the various components and tools of the ***SMARTFIRE*** environment. The advanced tutorials also cover some of the newest and research features of the ***SMARTFIRE*** Environment such as the ***SMARTFIRE*** Scenario Designer (for CAD import) and the prototype Parallel MPI version of the ***SMARTFIRE*** CFD Engine.

25.2 *TUTORIAL 1*

25.2.1 OVERVIEW

This tutorial takes the user through the simulation of a simple room fire case using an existing saved case specification. The files needed to run this tutorial are included in the installation of ***SMARTFIRE*** and will be found in the user's "work" directory. The case covered in this tutorial is for a centrally located fire in a single compartment (of dimensions of 2.8m x 2.8m x 2.2m height with a 0.74m wide x 1.83m high door). The fire is represented by a gas burner of dimensions 0.3m x 0.3m x 0.3m height with a constant heat release rate of 62.9kW. The walls are assumed to use a turbulent wall layer function for the calculation of heat loss. This case corresponds to one of the Steckler room fire experiments [1, 14, 15].

25.2.2 STEP 1: LOADING AN EXISTING GEOMETRY SPECIFICATION

Run the ***SMARTFIRE*** case specification tool. Once the graphical user interface has opened, use the menu bar at the top of the window to select [File] and then select [Open]. The case to be loaded for this tutorial is in a directory called "A74_case" within the ***SMARTFIRE*** work directory. Use the file browser to go to this directory and select the file called "*a74_case.smf*". This contains all of the geometric and physical properties for the specification of the case described above. Once the file has been selected and loaded, the viewer will show a 3D view of the room, together with all the important physical and geometric features, such as any vents, obstacles (if defined) and fires (See Figure 25-1).

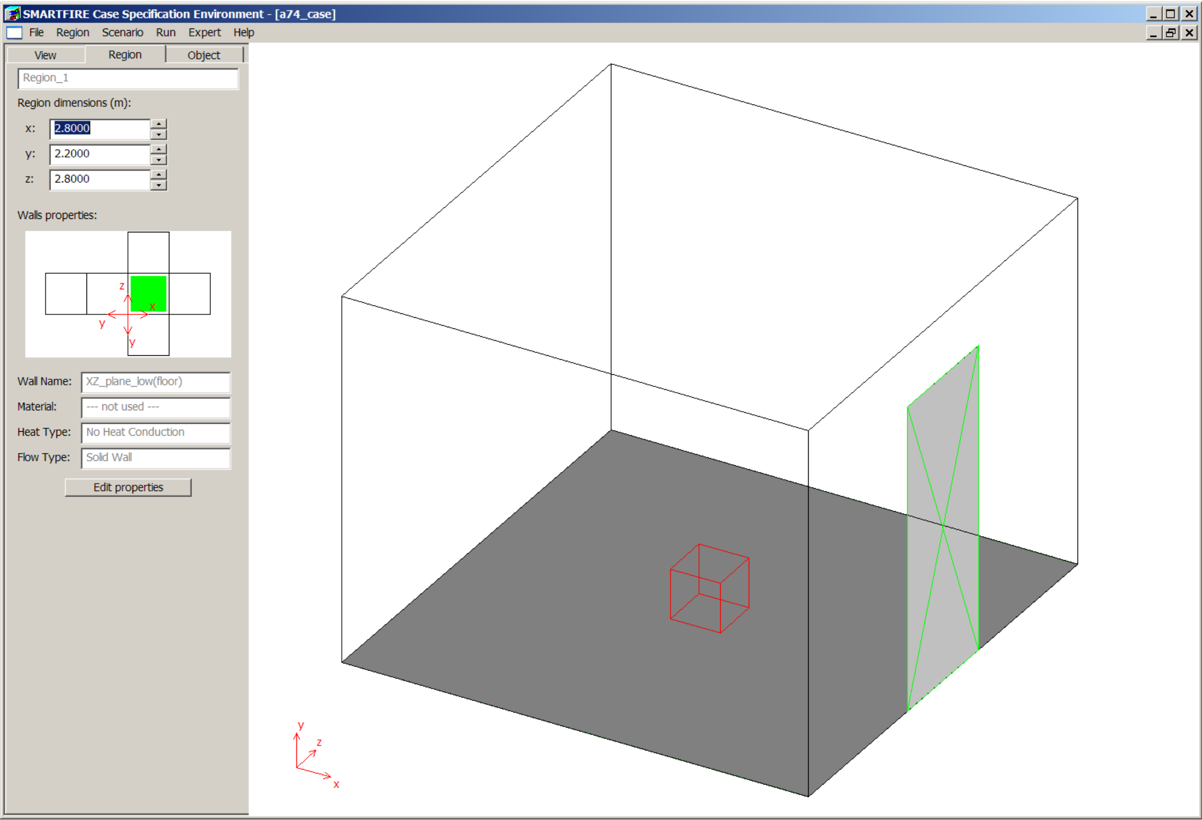

Figure 25-1 Specification environment showing the "A74_case" geometry.

Once you have opened the file, try experimenting with the viewing options. To start, you can select the [View] tab that allows you to change the way the viewer displays the geometry. You can try to rotate the geometry around the various axes, or chose to view a plan-view of a particular direction. It is recommended that you experiment with this and try to familiarize yourself with the various views offered. Additionally, you may decide to zoom in and change which items are visible at any time. Finally, you can get back to the original rotation by clicking on some of the pre-fixed views available. These view options are useful for checking the alignment of objects and checking the correctness of your geometry.
The middle tab is the [Region] tab. This allows you to change the dimensions of the room as well as the wall-materials and names of the walls. Press the button [Edit properties] to choose another material for a wall after using the *"unfolded room"* area to select the current wall. The current wall (including floor and ceiling) will be displayed with a solid grey fill.

The right tab is the [Object] tab. This allows you to add objects to the current region or to change the properties of any objects already created. You may like to experiment with moving and resizing the vent and fire. Additionally, you may want to change the physical properties of the fire by selecting it (by left-clicking on it with the mouse) as the current object and clicking on the object [Properties] button. An important point to note here is that only one object within the region will be current at one time and all of the buttons on the [Object] panel will cause an appropriate response for (and be related to editing) the current object. The current object is always displayed with grey shaded surfaces. You will notice that objects are drawn using coloured lines joining their vertices. The colours represent the object type. E.g. vents are green, fires are red and obstacles are blue.

25.2.3 STEP 2: CREATING A MESH FOR THE SIMULATION

Once you are satisfied with the geometry and physics handling of the case you need to run the mesh creation system. This is a combination of automated meshing tool (a "friendly expert" to help you perform the task of meshing) and a manual mesh editing tool. These tools are seamlessly embedded into the ***SMARTFIRE*** system. The automated meshing system will take some of the decisions that are needed to set up an appropriate computational mesh for this geometry and the requested cell budget.

To run the automated meshing tool from the main menu, choose the [Run] option, and then select [Create Mesh]. This will open the mesh creation tool with its own user interface.

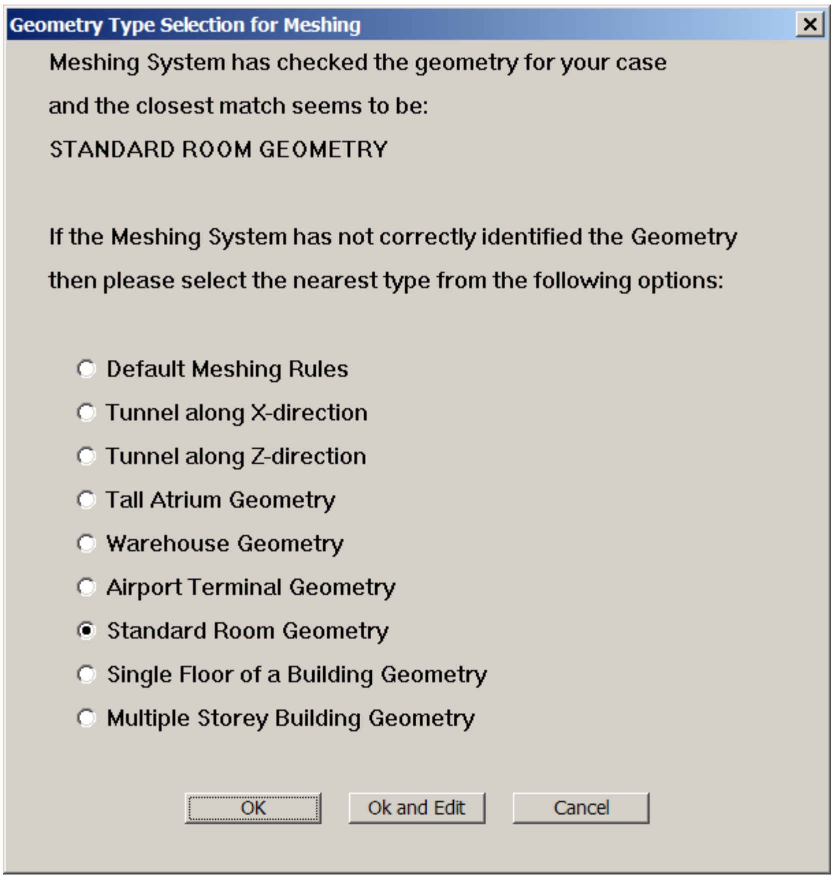

Figure 25-2 – Geometry type Meshing Rules selection dialog

The automated meshing tool will first determine if the current geometry already has an existing mesh previously created. If one is available then the user will be given the option to load the existing mesh or to create a new one. It is assumed that no existing mesh is available or that the user will create a new mesh. The user is then asked to select the geometry type meshing rules they wish to use with the dialog in Figure 25-2 above. For many scenarios, the appropriate rules are normally automatically selected, but the user may wish to override ***SMARTFIRE***'s suggestion if it does not seem to be the most appropriate type of geometry.

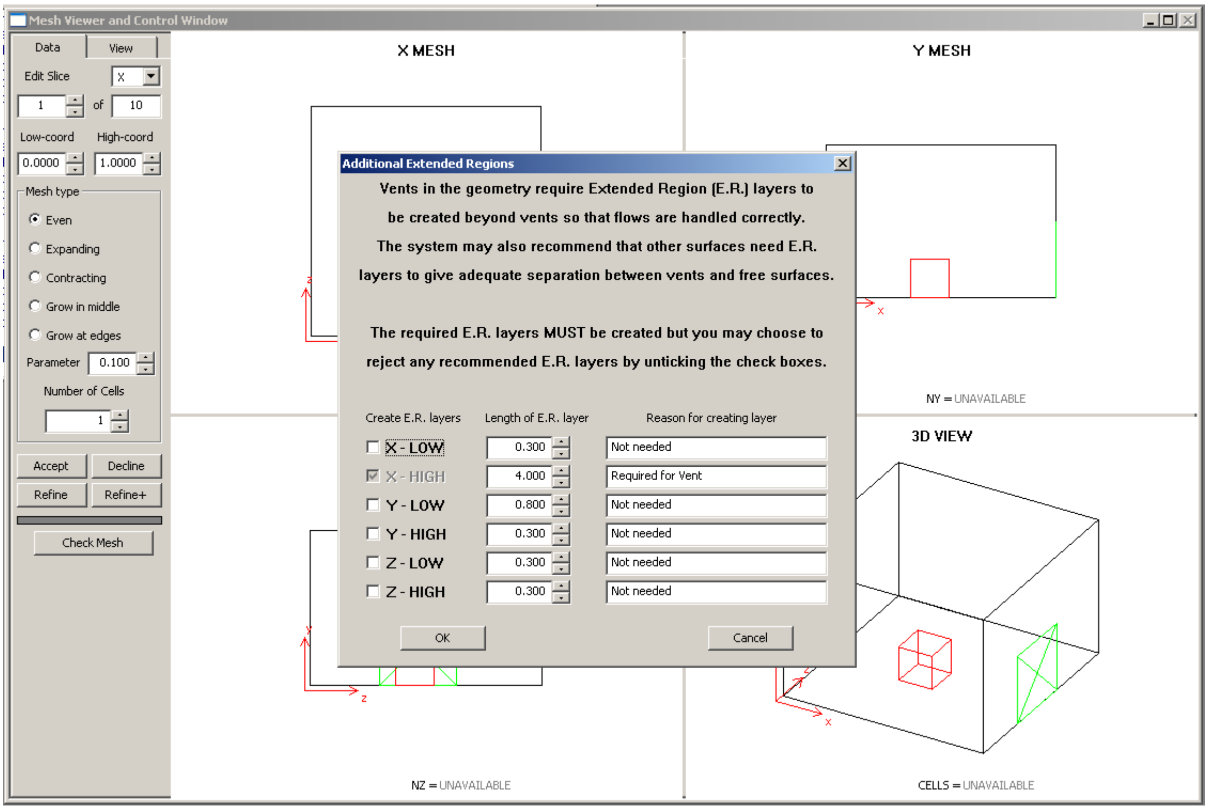

Figure 25-3 Additional Extended Region Dialog

The meshing tool will then analyse, using the chosen rule set, the geometry and may present a "Additional Extended Regions" window (See Figure 25-3). Extended regions are used to ensure the flow is correctly developed within the region of interest and the flow will not adversely affected by the outlet boundary conditions at the edge of the extended regions. The user does not normally need to adjust the selection of Extended Regions.

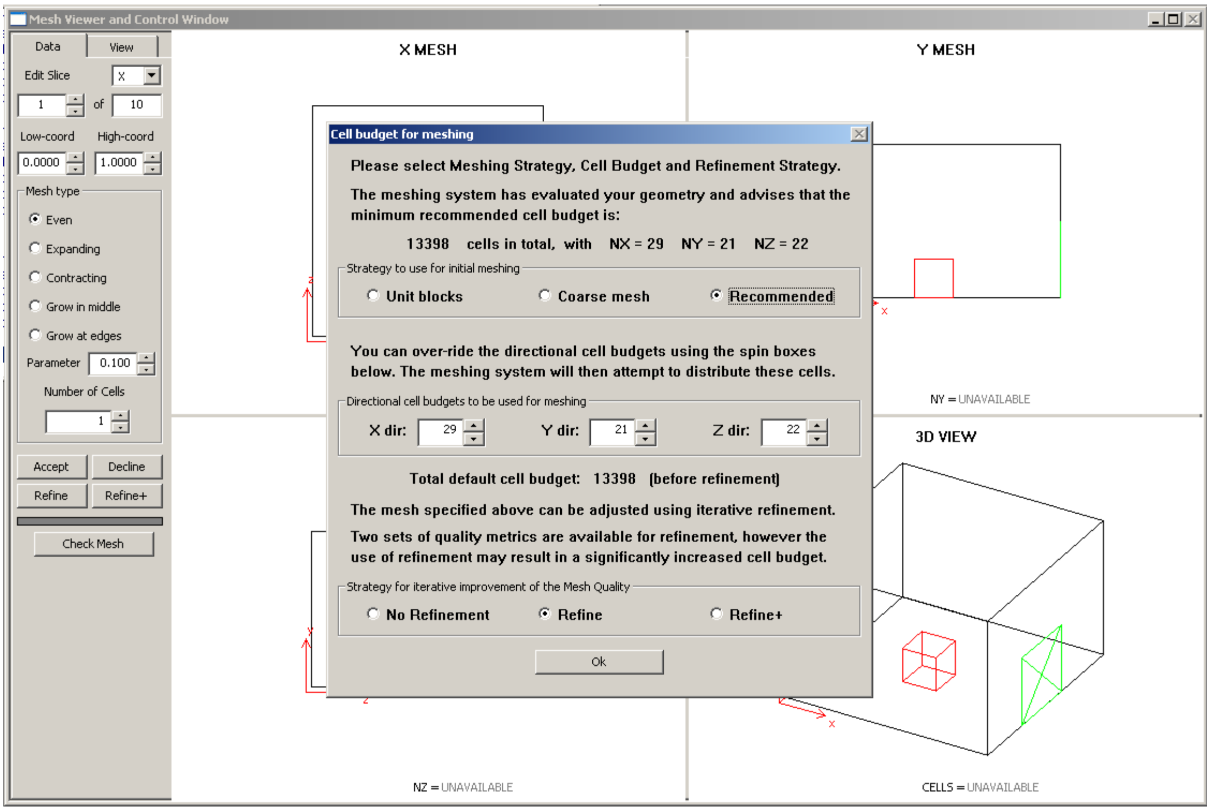

Figure 25-4 Meshing tool showing selection of recommended cell budget.

Once the meshing tool has loaded the case and considered the geometry, you will be asked to specify how fine a grid you wish to use in the simulation (see Figure 25-4). The automated meshing system will recommend a cell budget based on meshing expertise and the complexity of your geometry BUT it is up to you to decide if you want to use the recommended cell budget. Alternatively you can make the automated meshing tool try to produce the best mesh it can with a cell budget of your own choosing. Remember that a more refined grid will generally mean it is possible to obtain more accurate results but it will take longer to reach the solution. The automated meshing system will suggest a cell budget that is a reasonable compromise between accuracy and simulation time. You may accept the recommended value of cell budget by simply clicking on the [OK] button. This is likely to give a reasonable quality mesh using a fairly modest overall cell budget. The default refinement option should be used. When the meshing system finishes creating a mesh specification, the Mesh viewer window will be updated to show the mesh that has been created. The default mode of the Mesh viewer is for four views being the X-mesh, the Y-mesh, the Z-mesh and the 3D domain view. (See Figure 25-5). You will find that many of the grid lines actually correspond to the edges of physical objects (such as the edge of the fire or edge of a vent). The lines are coloured as follows mauve = block model of the domain with edges on identified sub-region boundaries, light-blue = internal mesh line between (block boundaries). The vents are coloured green and any fires are red. Obstacles are dark-blue.

Expert users may wish to use the [Coarse mesh] budget option to allow greater ease of creating a mesh manually. This option makes sure that all objects have at least the minimum number of required internal cells for their correct operation but only puts a single cell in non-critical blocks. The user can then easily use the interactive manual mesh editing tool to add cells as required to each of the under-populated blocks.

If you are not satisfied with the mesh that the automated meshing tool has created then you may press the [Decline] button (to discard the current mesh) and specify a larger (or smaller) cell budget for re-meshing. Alternatively you may use the interactive editing mode to add (or subtract) cells from mesh blocks or change the distribution of cells.

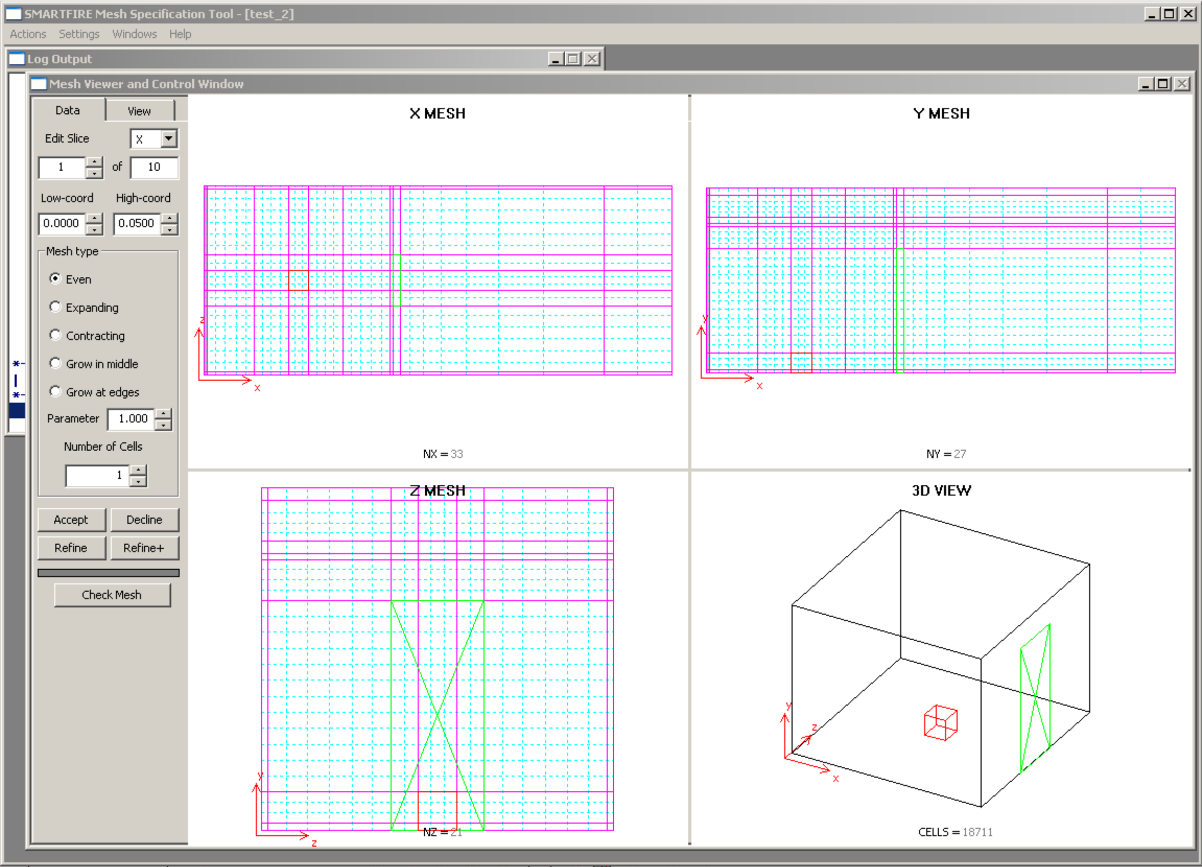

Figure 25-5 Meshing tool showing a four views of the mesh and geometry.

There is also an option to evaluate the quality of the created mesh. This quality check can be used to determine if the cell aspect ratios (between the directions in a cell and between adjacent cells) are acceptable.

Finally, when you are happy with the mesh, you need to press the [Accept] button to allow this mesh to be converted-to and saved-as a set of specification files for the CFD Engine to read. Pressing the [Accept] button will also cause the automated meshing tool to close and control will then return to the ***SMARTFIRE*** Case Specification Environment.

25.2.4 STEP 3: RUNNING THE CFD ENGINE

The final stage involves performing the numerical simulation itself using the CFD engine component of ***SMARTFIRE***. To run the CFD engine select on the main menu item [Run] and the [Run CFD Engine] option. This will launch the numerical CFD engine that will automatically load the case specification and mesh that you have just created. You may have to be a little patient as this stage involves a considerable amount of file parsing, memory allocation and initialisation. Eventually the user interface for the CFD engine will appear completely (See Figure 25-6). It should be noted that if you are using a fairly low resolution display then the CFD engine user-interface may display itself with more compact windows and a slightly different layout (e.g. more windows initially closed) in order to create an easily usable display.

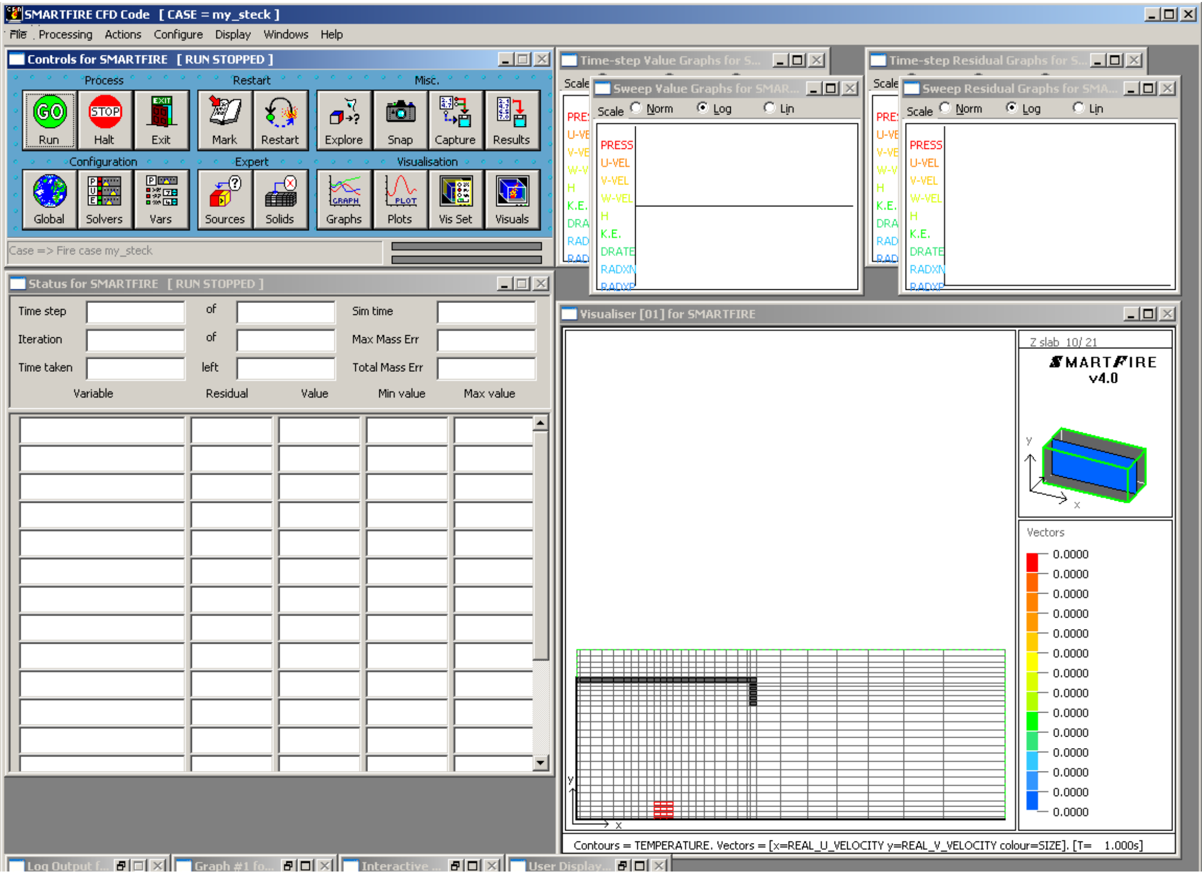

Figure 25-6 *SMARTFIRE* CFD Engine user interface at start up.

In order to start the numerical simulation, you need to press the [Run] button, marked with a GREEN (GO) icon. This will start the simulation process. To halt the simulation at any time, press the [Halt] button marked by a RED (STOP) icon. You can also terminate the simulation at any point by pressing the [Exit] button (showing a little door icon). There are a number of extra buttons and other controls that allow experts to control the configuration of the solution process. However, non-experts are not recommended to make any changes to the control settings unless such changes are recommended in the user guide.

The graphical windows of the user interface present various views of the data and the status of the simulation during the simulation process. The things to look for are:

1) The residual graphs that are indicators of the convergence of the solved and calculated variables of the numerical simulation process (Top right).

2) The monitor values and variable residuals (current solution error states) of various important variables are shown in the status window (Bottom left).

3) The emerging vector flow and temperature contour patterns for the particular selected slice of the room in the visualizer window (Bottom right).

4) The data ranges for each variable displayed in the status window (Bottom left).

5) The control window has progress bars in its bottom right corner (below the visualisation buttons). These bars indicate solution progress. The upper bar is filled once every sweep whilst the lower bar is filled once for the whole configured simulation (Top left).

6) The status window has displays indicating the sweep number and time step number (only for transient simulations) to indicate the current stage of processing (Bottom left).

7) The status window has estimates for the CPU time taken and remaining. These are only estimates but can give a reasonable approximation of the expected duration of a simulation (Bottom left).

8) A key feature of ***SMARTFIRE*** is the access to save a bookmark and restart from saved bookmarks at any time. The control button labelled [Mark] will drop a bookmark of the current stage of the solution into a database for this case. The button labelled [Restart] allows a previous bookmark state to be loaded as if subsequent processing had not happened. This can be invaluable for problematic simulations that need expert solution control or simply for saving data for future examination (Top left).

9) There is a control button labelled [Plots] that allows you to define line graphs through the data. These plot line graphs are updated as the solution progresses (Top left).

If you need to change the particular viewing slice through the room then you can select the [Visuals] button. This will open the visual configuration window (See Figure 25-7).

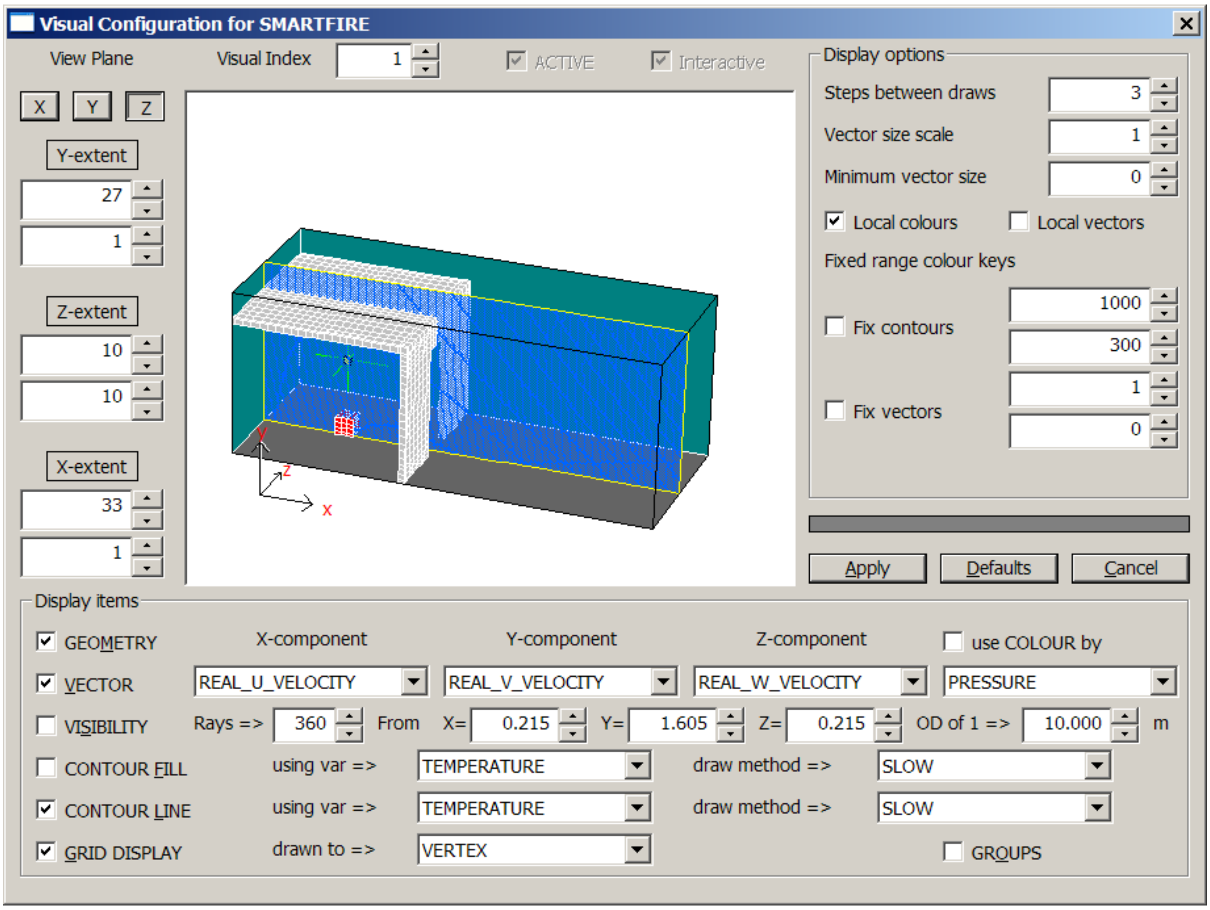

Figure 25-7 Visual Configuration showing slice and display options.

This dialogue allows you to select a particular slice through the room (and the nature of the visualisation to be displayed). For the case shown in this tutorial, you should select the 11th “X against Y” slab on the Z direction (referred to as Z=11). This should allow you to see the fire, the door and the emerging fire plume that first goes up to the ceiling and then leans over towards the back of the room because of cold air rushing into through the bottom of the door. An interesting alternative is to view a slice through the first slab on the y axis (height). This will pinpoint the fire within the room.

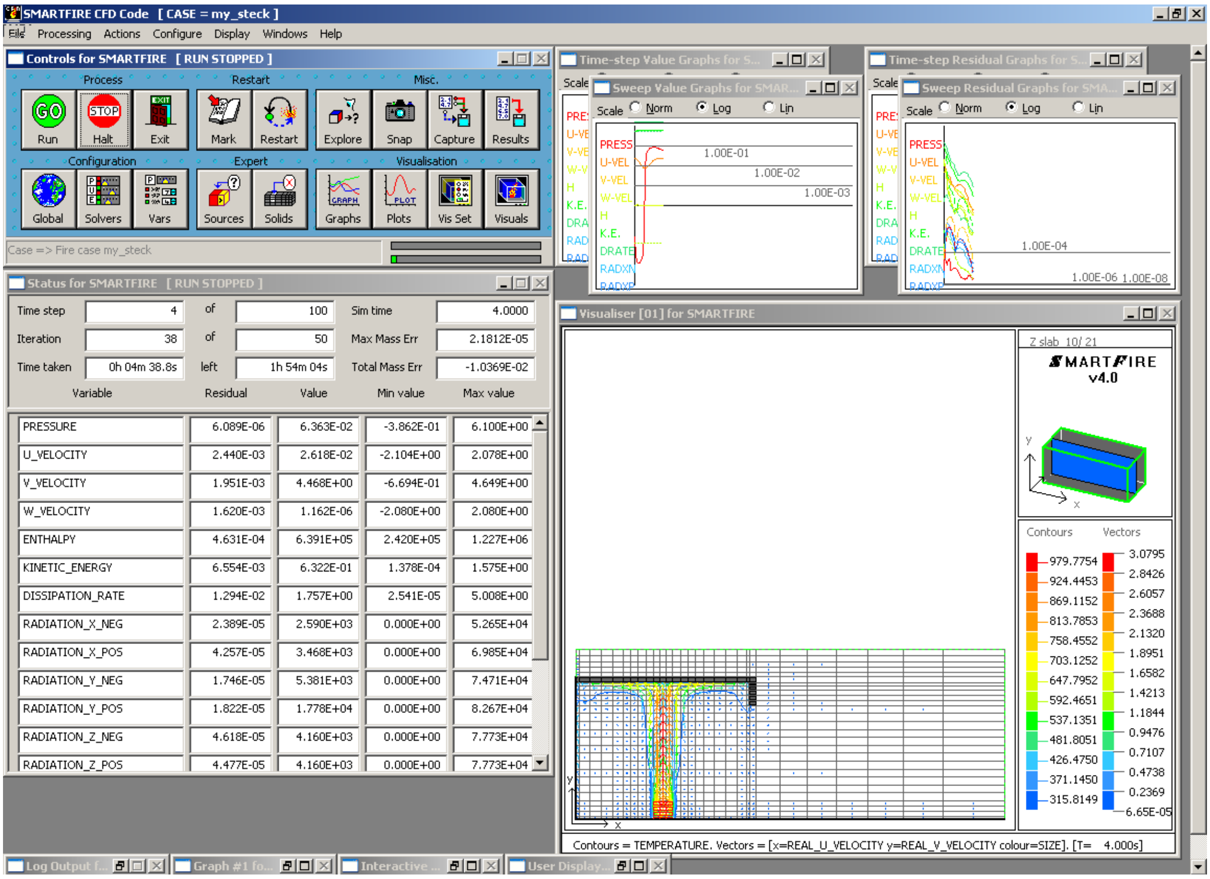

Figure 25-8 Visualisation of temperature and velocities for mid Z-slice.

Other features that you may control, using the visual configuration window, are the variables used for the contour and vector plots and the types of plots used. Try to experiment with some of these settings and examine the pull down lists to see what other variables are available for visualisation. See how changing the settings affects the visual display.

25.2.5 STEP 4: EXITING THE CFD ENGINE

To exit the code, you first need to exit the CFD engine interface using the [Exit] button. On normal termination, the ***SMARTFIRE*** CFD engine will save a number of files that can be used for further visual post-processing (abnormal termination will not save any files and is encountered when the main window [X] button is pressed). Finally the CFD engine user interface will close and you will get back to the original geometry set-up tool. If you want to save any changes you have made to the fire modelling case, select the [File] item and then the [Save] option from the main menu.

You may like to try saving your work as a new case because it is generally necessary to create a new case for each new geometry and simulation you want to run. To save the scenario specification as a new case, you should choose the [File] item from the main menu and then the [Save As] option, and enter a new case name in the file browser window. The case specification environment will create a directory folder, of the same name, to contain all files relating to your new case and it will save the "new_case_name.smf" file in this directory.

It should be noted that the case directory (for a completed simulation) will contain many files that have been created during this exercise. Some of these files can be used for graphical post-processing and for re-starting this simulation from the stage at which it was ended. Also there may be data capture files that were saved during your simulation.

You have just completed a fire field modelling simulation, using ***SMARTFIRE***.

This is the end of tutorial 1.

25.3 *TUTORIAL 2*

25.3.1 OVERVIEW

This tutorial takes the user, step by step, through the specification and modelling of a simple room fire case using the specification environment. The case covered in this tutorial is a centrally located fire in a single compartment of dimensions 2.8m x 2.8m x 2.2m height with a 0.74m wide x 1.83m high door. The fire is represented by a gas burner of dimensions 0.3m x 0.3m x 0.3m height with a linear ramped heat release curve up to 62.9kW in 20 seconds. The six-flux radiation model is used to represent radiative heat transfer. The walls are assumed to use a turbulent wall layer heat transfer function. This case corresponds to one of the Steckler room fire experiments [1, 2, 3]. This case is essentially the same as the case in tutorial 1 but it must be specified manually using the case specification environment.

25.3.2 STEP 1: STARTING THE CASE SPECIFICATION TOOL

Run the ***SMARTFIRE*** case specification tool. Once the graphical interface has opened, use "*Region*" tab (if not already selected) to show the region editor panel. You will be shown a visual depiction of an empty, default sized region with no geometry objects defined (See Figure 25-9).

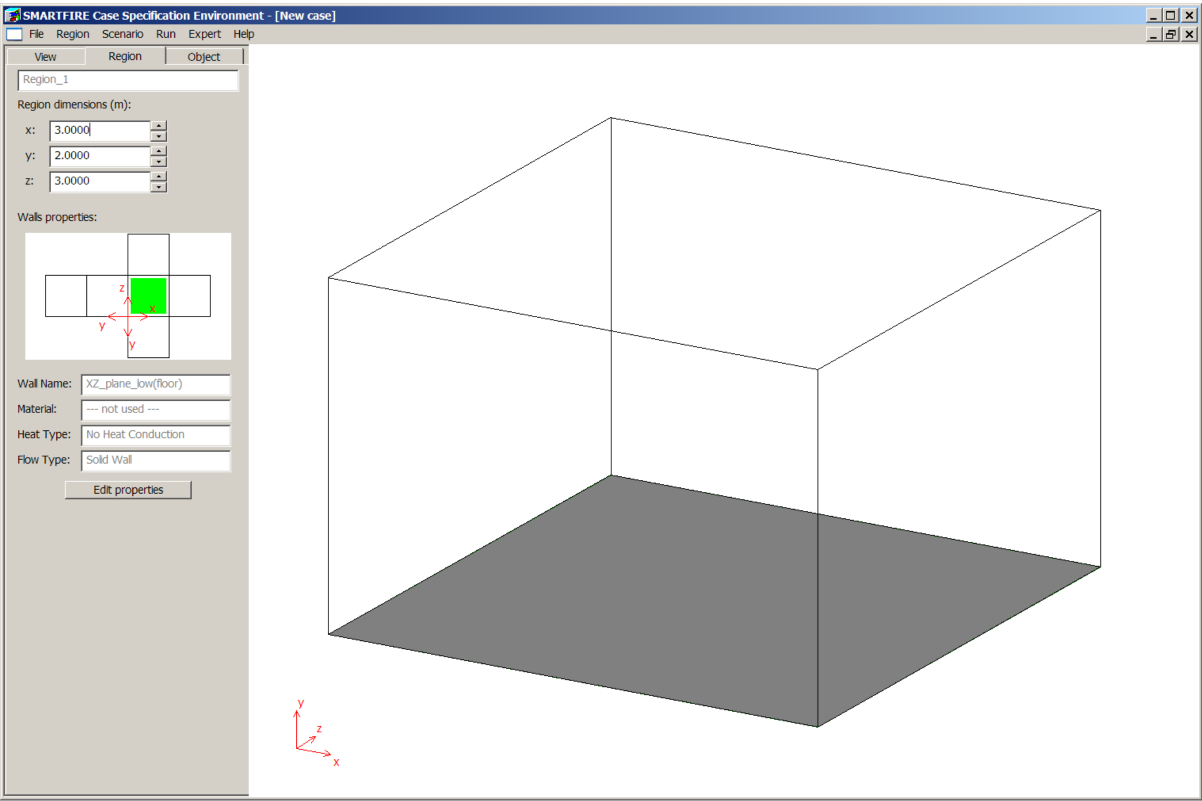

Figure 25-9 Specification tool showing an empty default initial region.

Note the wall surface selection tool (unfolded box) on the left hand side of the display and the corresponding shaded surface in the region display.

25.3.3 STEP 2: CREATING THE DOMAIN

Enter the dimensions of the region in metres using the three spin boxes (See Figure 25-10). You can select on the spin increment and decrement buttons to change the displayed size by steps of 0.1m or you can double-click the value and type a new one in directly and press the enter button to confirm your input. As specified in the case description, the dimensions are set as follows: x-size = 2.8m, y-size = 2.2m and z-size = 2.8m. You do NOT need to change the wall behaviour for any of the region walls because the default behaviour is acceptable for this case.

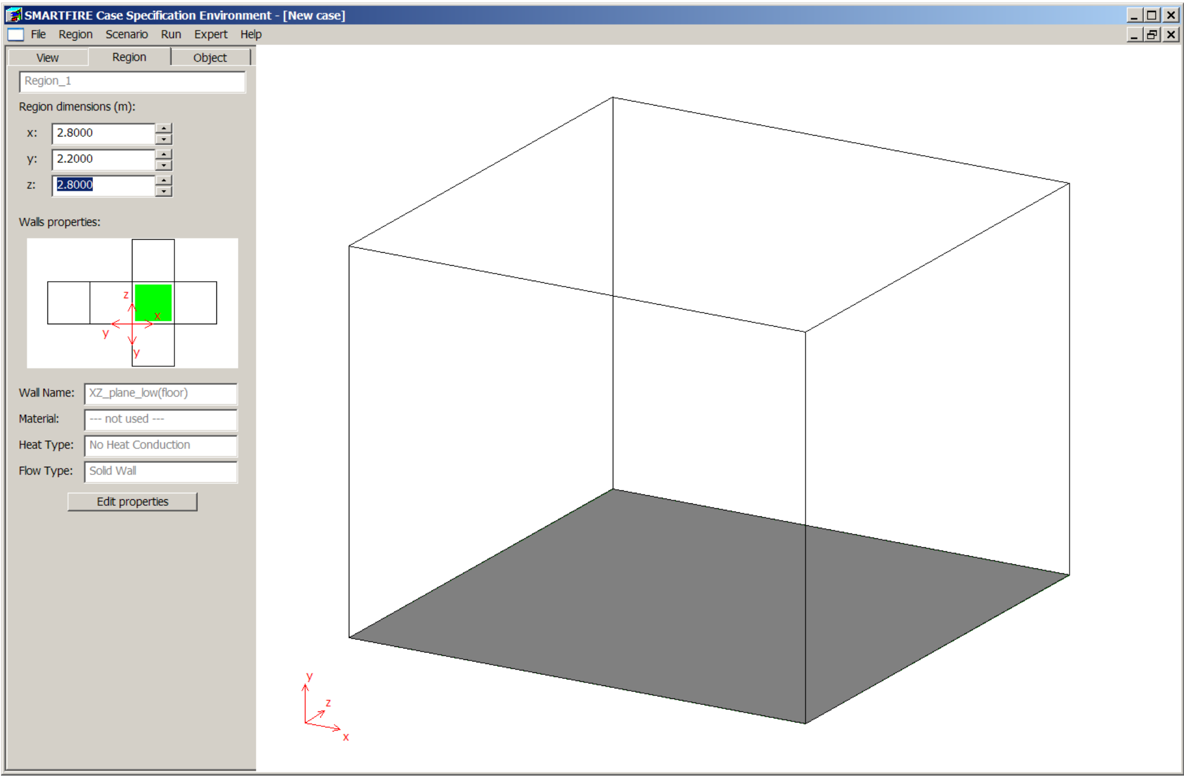

Figure 25-10 Specification tool showing entry of correct region sizes.

25.3.4 STEP 3: CREATING GEOMETRY OBJECTS

Once the region has been sized correctly you can create all of the geometry objects (fires, obstacles and vents) that are required for the simulation. In this tutorial you will need to create a fire and a vent.

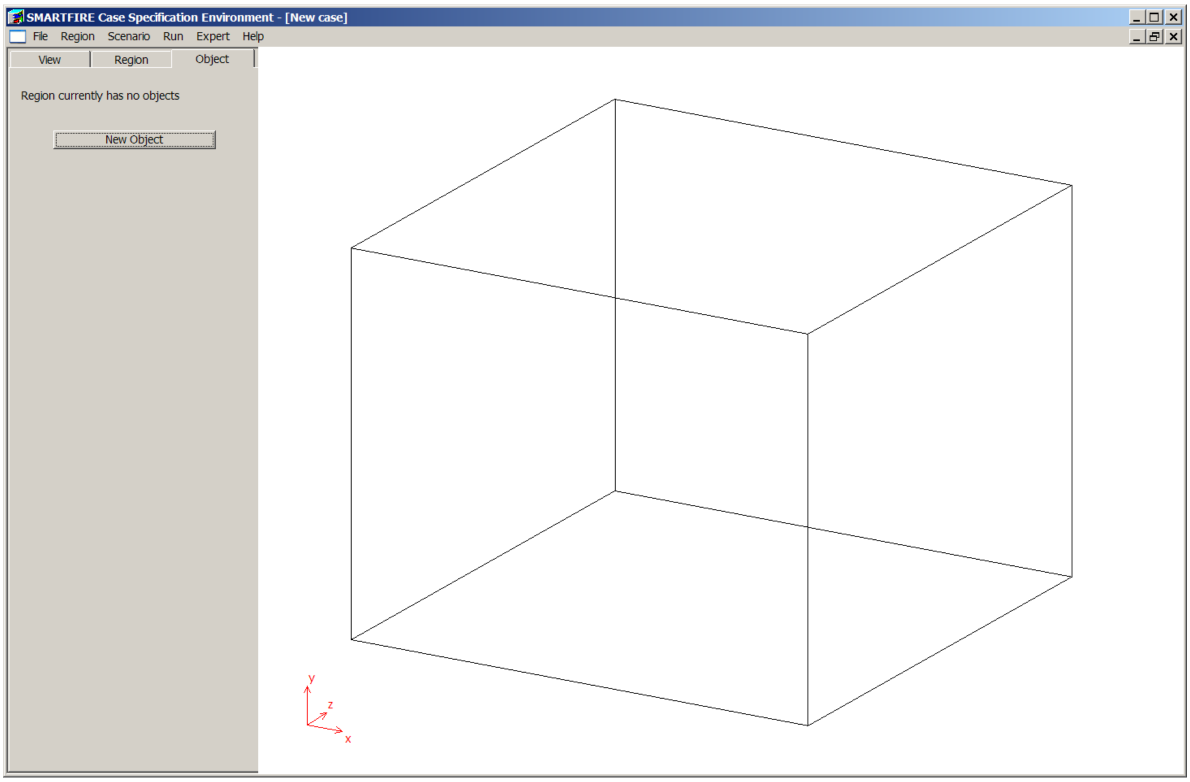

Figure 25-11 Specification tool showing object editor panel.

Select the "*Object*" tab to display the object editor panel (See Figure 25-11). There are initially no objects defined so the panel only displays a "*New Object*" button. Select the button to open the new object menu. This menu presents a list of all of the object types available. Select the "*VENT*" object and press the "*Add*" button (See Figure 25-12).

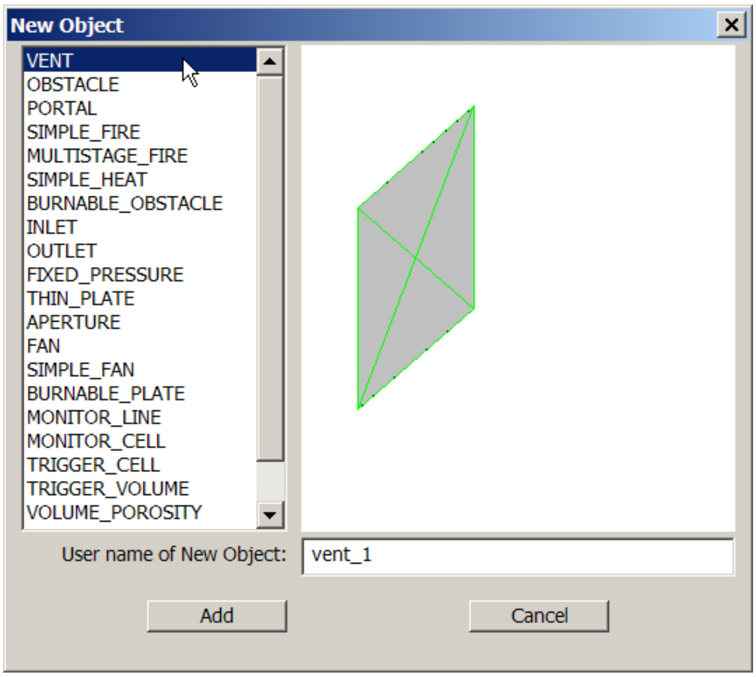

Figure 25-12 New object selection dialogue used to create a new VENT.

The dialogue window will close and a default sized rectangular vent will be added to the region display at a default location (See Figure 25-13). The object editor panel will modify itself to match the currently selected object type (in this case the vent you have just created). The current object will always be displayed using its own typed colour key for the edges and shaded GREY faces. Non current objects do not have GREY shaded faces.

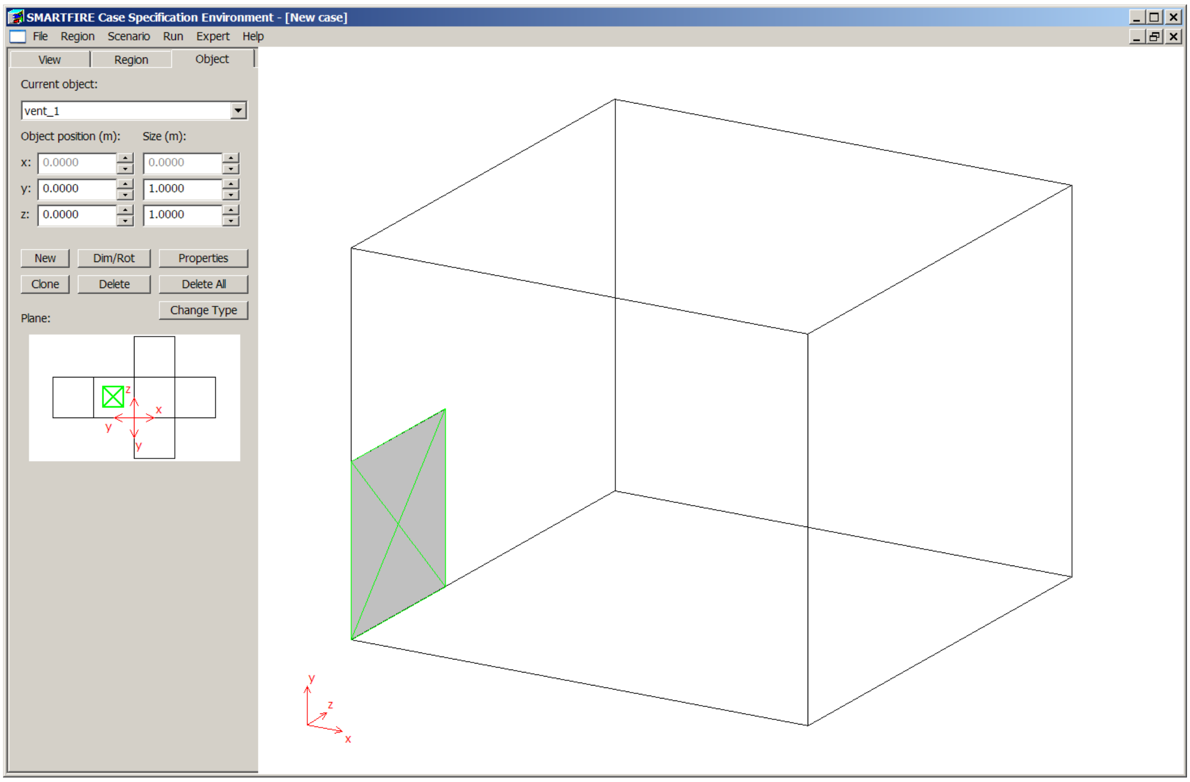

Figure 25-13 Specification tool showing new VENT at default location.

In this simulation it is required that the vent (doorway) has a width = 0.74m and height = 1.83m and be centrally located on one of the walls. Arbitrarily, the vent will be placed on the high-x surface of the region. Use the "*unfolded region*" display to choose a region face so that the vent appears on the high-x surface of the region (See Figure 25-14).

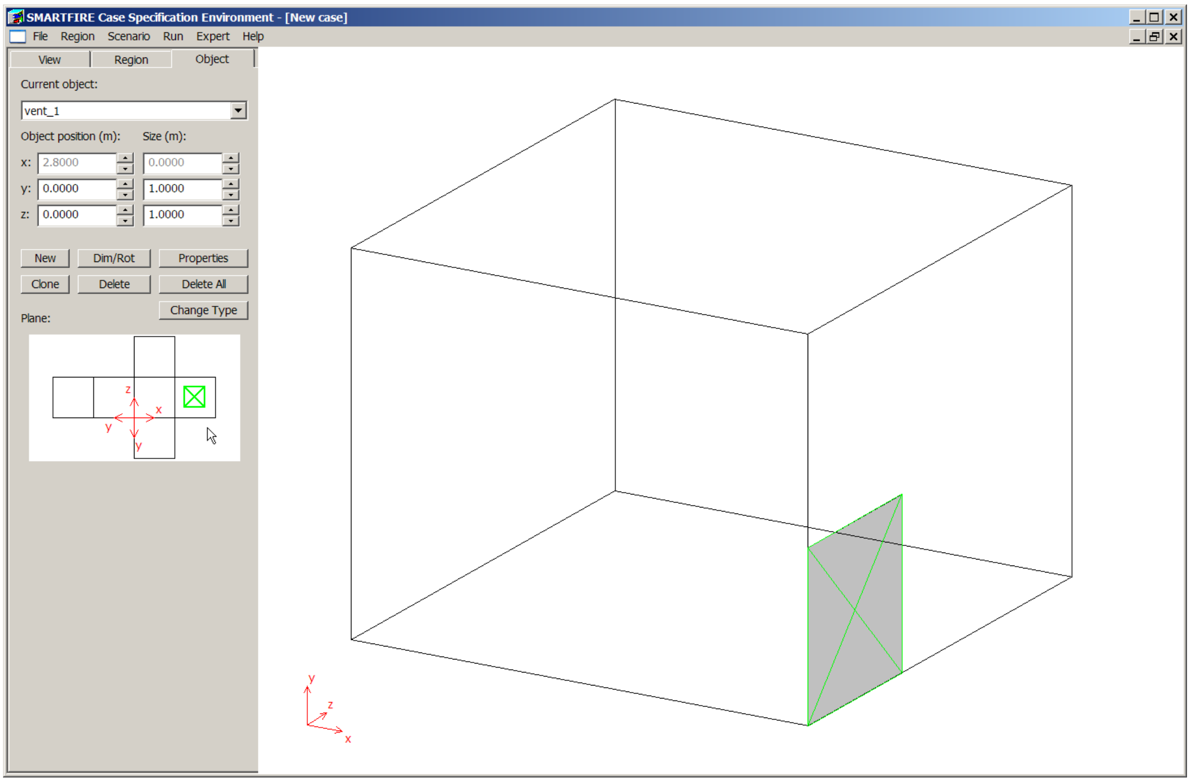

Figure 25-14 Specification tool using the "unfolded" plane selector.

Enter the size and location of the vent in the spin boxes on the object editor panel (See Figure 25-15). Since this is a 2 Dimensional object only the relevant spin boxes will be enabled for entering sizes and locations. In the case of the doorway specified in the case description you will need to enter the following values provided that the vent is on the high-x surface: y-size = 1.83m, z-size = 0.74m and y-location = 0.0m and z-location = (2.8m / 2) - (0.74m / 2) = 1.03m. The vent will now be correctly sized and centrally located on the high-x surface of the domain.

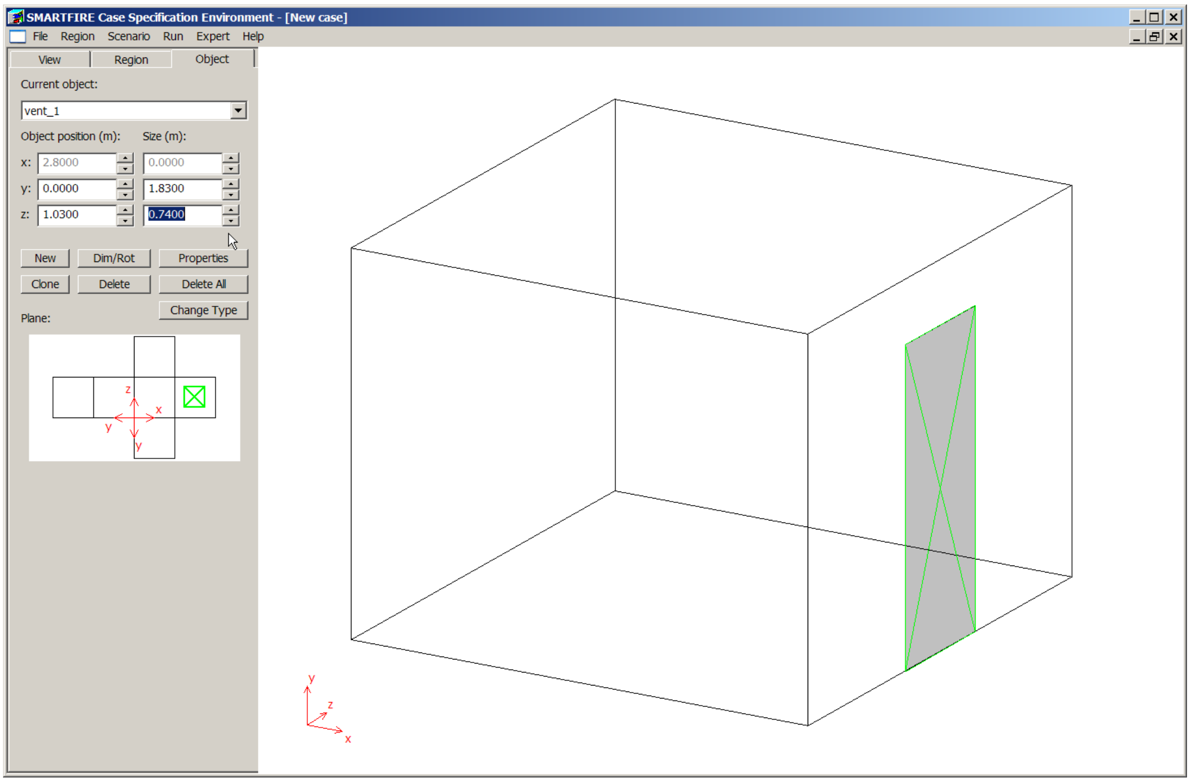

Figure 25-15 Specification tool showing correct VENT size and position.

The final object that needs to be created is the fire source. Select the button labelled "*New*" in order to choose another object type to add to the region. This time select the "*SIMPLE FIRE*" item from the list and press the "*Add*" button (See Figure 25-16).

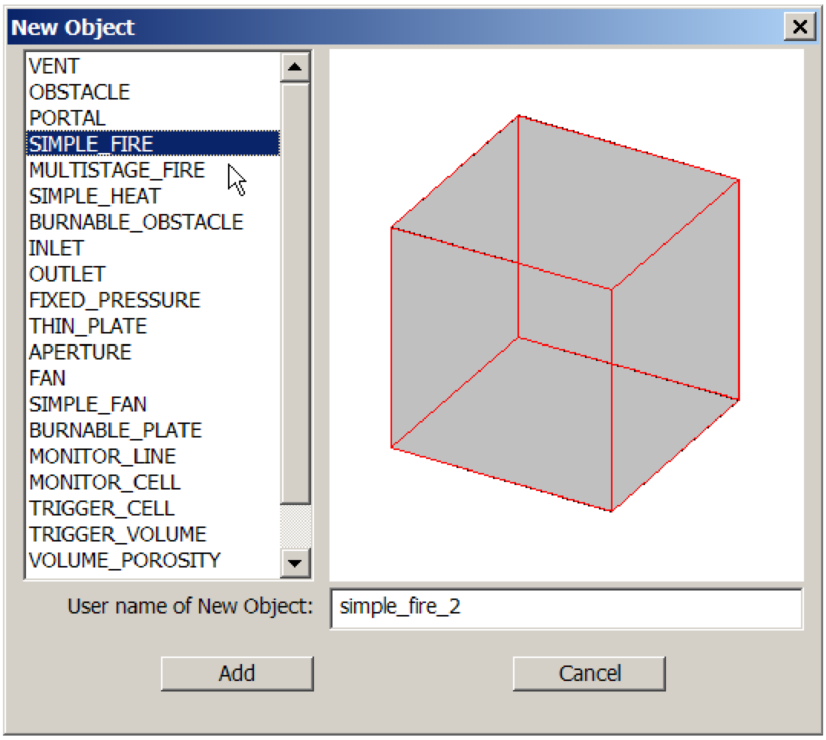

Figure 25-16 New object dialogue used to create a new SIMPLE_FIRE.

A default sized cube will placed at the low coordinate (0.0, 0.0, 0.0) location of the region and will be made current (GREY faces) (See Figure 25-17).

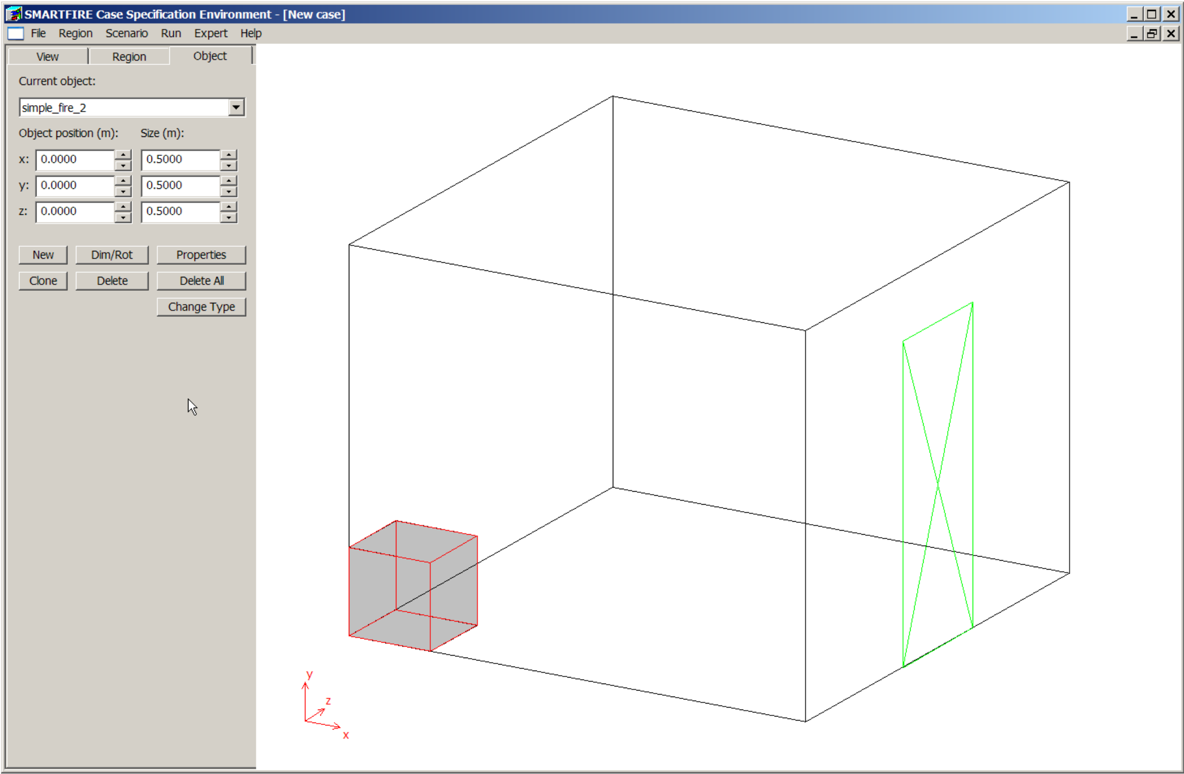

Figure 25-17 Specification tool showing default SIMPLE_FIRE.

Enter the sizes of this fire using the spin boxes. In this case the dimensions are x-size = 0.3m, y-size = 0.3m and z-size = 0.3m. The locations are x-location = (2.8m / 2) - (0.3m / 2) = 1.25m, y-location = 0.0m and z-location = (2.8m / 2) - (0.3m / 2) = 1.25m. Enter these values in the appropriate spin boxes and the fire will be centrally located and correctly sized (See Figure 25-18). The fire will have a default fire output curve specified (a fire with a constant Heat Release Rate of 50kW) which is not appropriate to the case specification. Select the button labelled "*Properties*" to access the fire properties editor window. It should be noted that the "*Properties*" button will always open an editor window appropriate to the type of the currently selected object and the values and settings displayed will be those of the current object.

The fire properties window allows you to choose a fire behaviour curve from a number of given or parametric heat release curves. In the case description, the fire is required to be linearly increasing with time for 20 seconds up to a final sustained value of 62.9kW (See Figure 25-19).

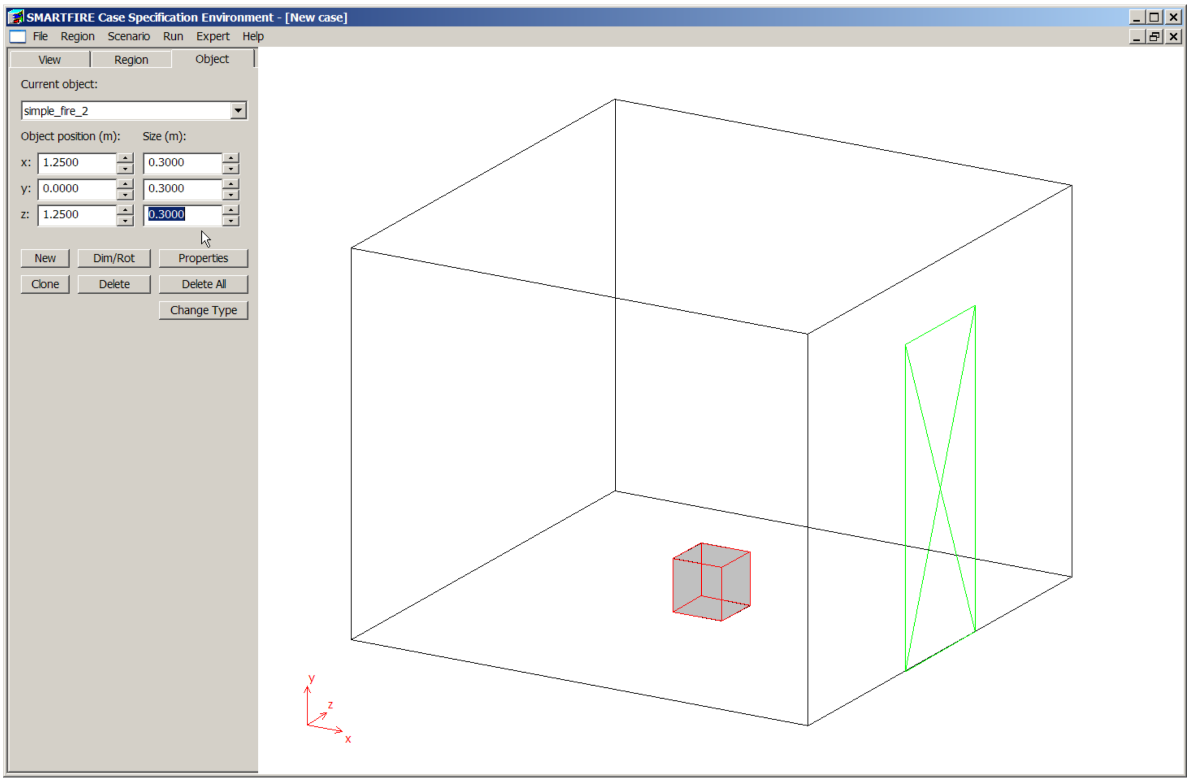

Figure 25-18 Specification tool showing FIRE position and size.

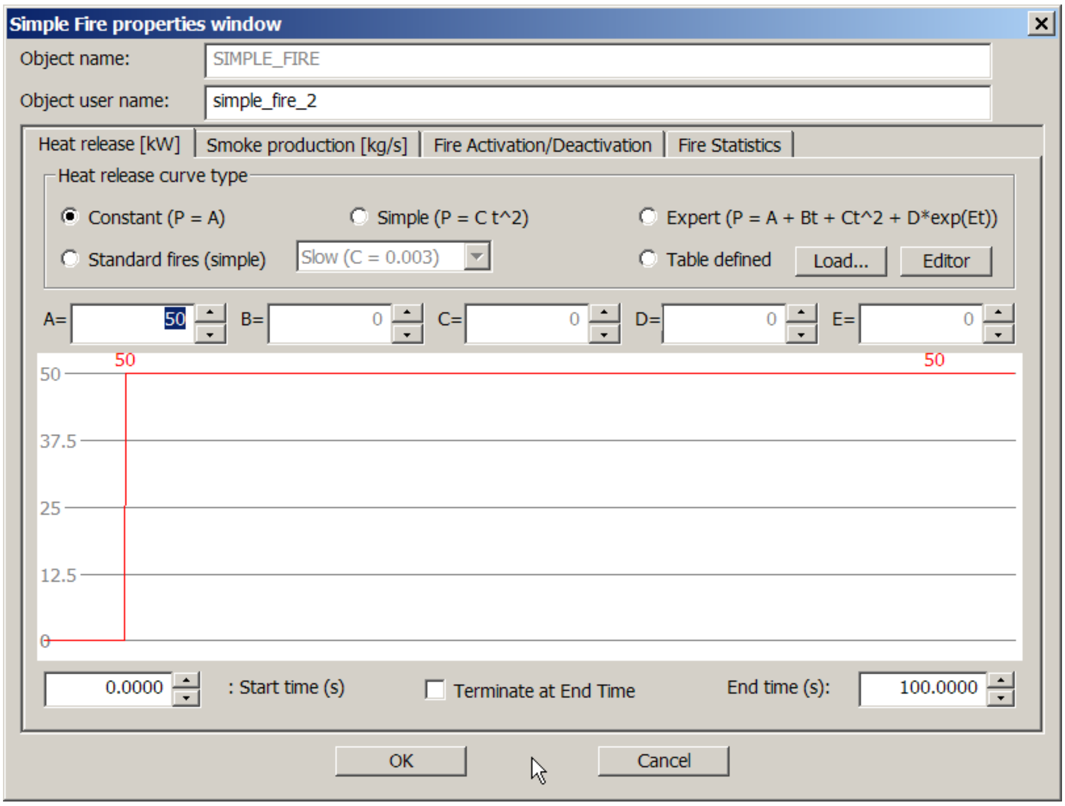

Figure 25-19 Fire properties dialogue showing default fire settings.

This "linear" type of fire growth corresponds to an "Expert" parametric curve with a coefficient for linear time as B = (62.9 / 20) = 3.145kW. Select the "Expert" radio button to enable the parametric curve and enter 3.145 into the B value spin box. All of the other spin box values should be set to zero. I.e. A = C = D = E = 0.0 (See Figure 25-20).

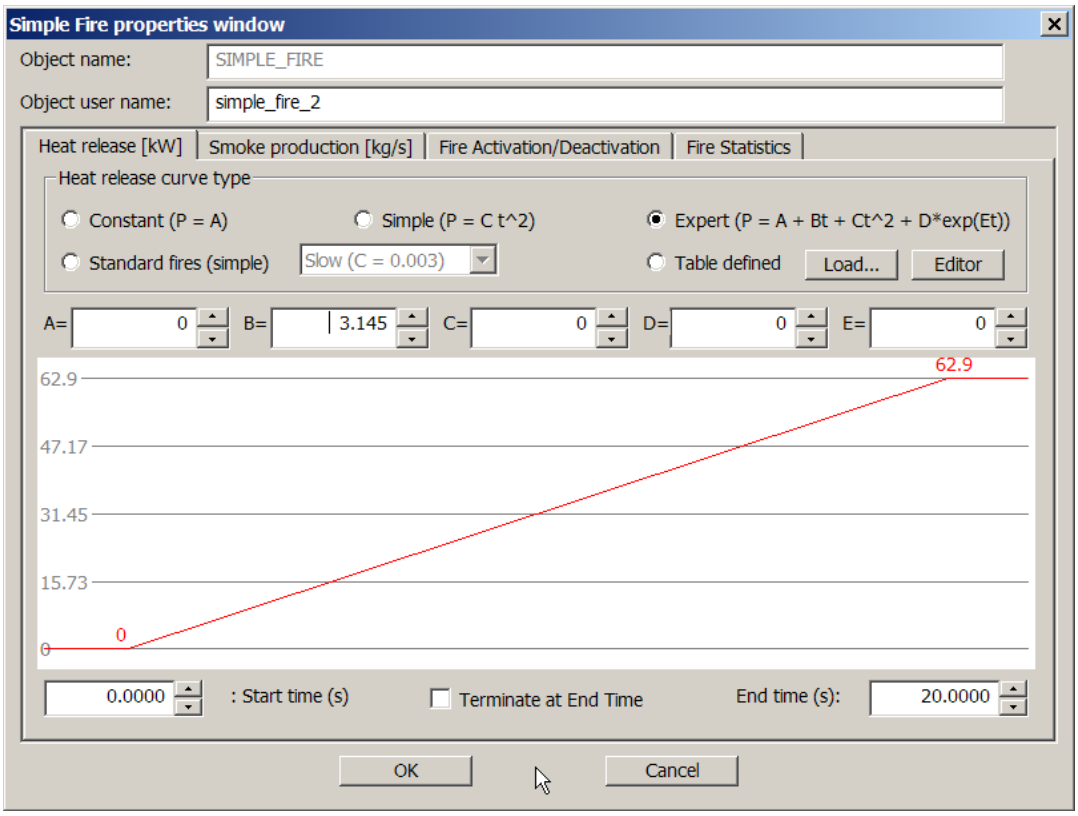

Figure 25-20 Fire properties showing a linear fire curve to 62.9kW in 20.0s.

Enter 0.0 in the Fire start time spin box and 20.0 in the Fire end time spin box. You should notice that the red power curve is indeed displayed as a linear ramp up from left to right and the upper right hand edge has a value of 62.9 displayed above it. This is the heat release curve required by this tutorial. Select the OK button to accept the settings and close the dialogue.

If you are confused about setting the specified fire curve, you may alternatively enter a simple constant fire of the correct power by selecting the "*Constant*" curve type and entering 62.9 into the A spin box (which will be the only setting active for a constant fire).

25.3.5 STEP 4: DEFINING THE PROBLEM TYPE

Select the "*Scenario*" menu from the main menu bar and choose the "*Problem Type*" item. This will open a configuration window that allows the simulation controls to be modified (See Figure 25-21). The Steckler A74 case was found to reach near steady state after 30 minutes BUT indicative results can be obtained after 3 minutes. In order to perform a reasonably short simulation it will be necessary to change the simulation time characteristics. An acceptable simulation can be performed using 18 time steps of 10 seconds per time step and 50 iterations per time step. Enter the following values to configure the simulation characteristics: Time step size = 10.0s, Number of time steps = 18 and Number of iterations per time step = 50. This will give (18 x 10) = 180 seconds of simulated time. I.e. 3 minutes.

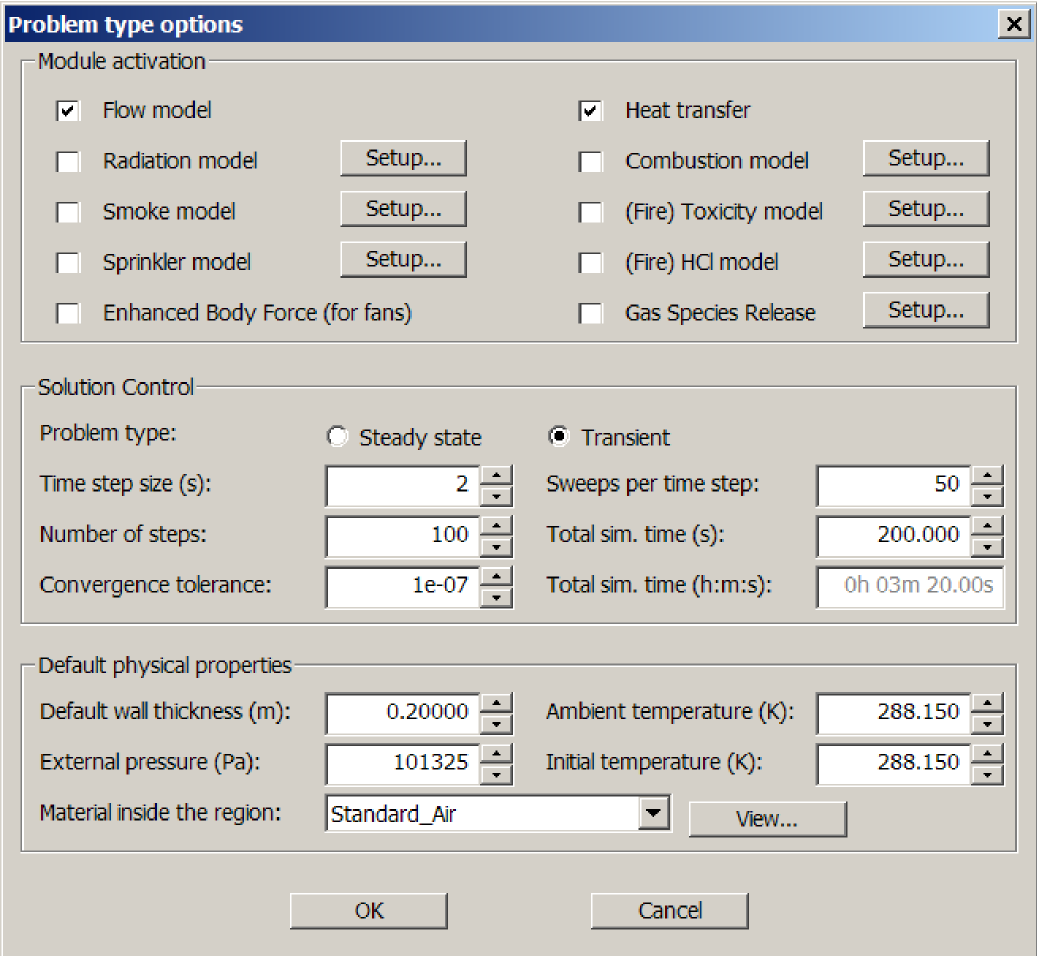

Figure 25-21 Expert Problem Type showing the type of simulation.

To enable radiation you should select the "Radiation model" check box. You may need to go to the "Setup…" for the radiation model to check that the correct type of radiation model is to be used. By default ***SMARTFIRE*** uses the Six-flux radiation model. None of the other parameters need to be changed from their default values but the user is advised to glance at the other global control items that can be configured in this window.

25.3.6 STEP 5: SAVING TO A NEW CASE DIRECTORY

Now that the simulation has been fully specified it is necessary to save the geometry and physics specification to a unique case directory where all of the other case-related simulation files will be created and stored. Select the main menu "*File*" option and choose the "*Save As...*" item. This will open a file browser that is viewing the contents of the ***SMARTFIRE*** work directory (See Figure 25-22). Enter a unique name for this simulation case. It is recommended that you choose a representative name that you can use to easily identify this case again in the future. For example you could enter the name "Tutorial_2" and then select the "*Save*" button. This will create a directory (folder) called Tutorial_2 and will save the specification file "Tutorial_2.smf" into the new directory. You should not create new case directories yourself since ***SMARTFIRE*** expects to do this whenever you save a new case.

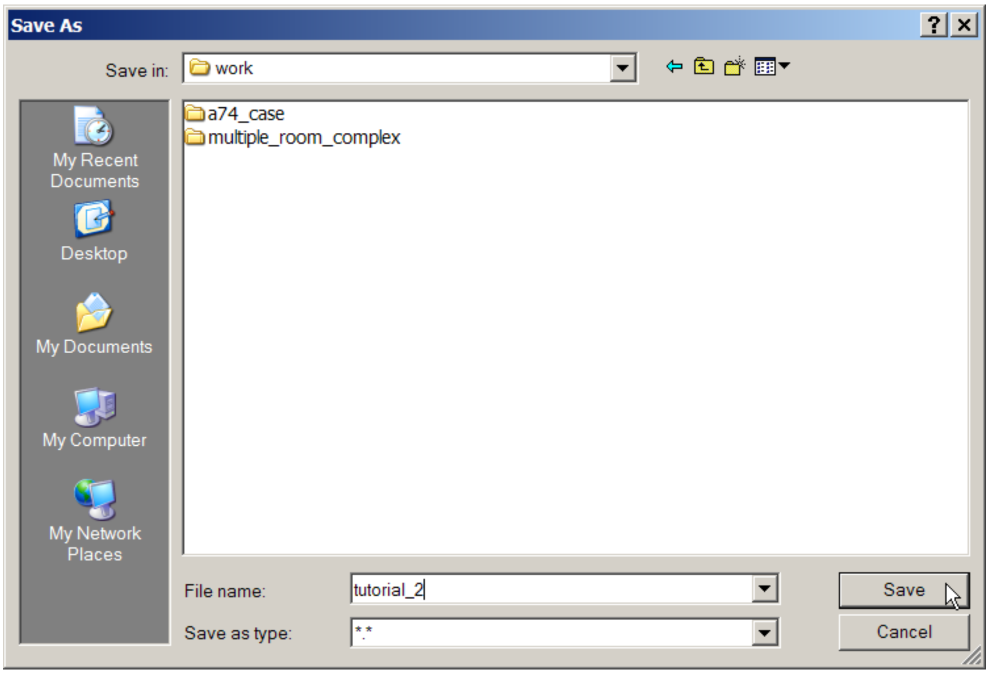

Figure 25-22 Using File "Save As..." to specify a new case name.

STEP 6: FINISHING SPECIFICATION AND STARTING CFD ENGINE

As with Tutorial 1 you are now ready to create a mesh specification, a specification script file and to simulate this case in the CFD engine. The steps to achieve this are described in more detail in Tutorial 1 but are enumerated here in succinct form for completeness.

- **Use the main menu "Run" and "Create Mesh" to run the mesh generation system.**
- **Choose a cell budget (as NX, NY and NZ) for simulating this case (it is recommended that, for this scenario, a total cell budget of 11000 or so cells gives a reasonable quality mesh).**
- **"Accept" the created mesh to create the geometry file and specification script - and this will also exit from the automated meshing tool.**
- **Start the CFD engine by selecting the main menu items "Run" and "Run CFD Engine".**
- **Wait for the CFD Engine to initialise.**
- **Start the CFD Engine using the "Run" button.**

At any stage of the processing you may stop the CFD engine, using the "Stop" button, in order to change the visualisation view, to explore the data, to define graphs or to modify the control parameters BUT you must wait for the current sweep to end (upper GREEN progress bar must have reached the right hand edge in the Control window).

When you have finished you should stop the simulation and press the Control panel "Exit" button to terminate the CFD engine.

This is the end of tutorial 2.

25.4 *TUTORIAL 3*

25.4.1 OVERVIEW

This tutorial takes the user, step by step, through the specification and modelling of a multiple room fire scenario using the specification environment. The case covered in this tutorial is an elongated dual compartment of dimensions 6.1m x 3.0m x 2.5m height with a 1.0m wide x 2.0m high door at one end of the long-axis. The region is divided equally into two compartments by a 0.1m thick partition half-way along the long axis with a doorway of the same location and dimensions as the end vent (door). There is a centrally located 1.0m x 1.0m window on the side wall of the compartment without an external door. The fire is centrally located in the compartment that does not have an external door and is represented by a gas burner of dimensions 0.5m x 0.5m x 1.0m height with a constant heat release rate of 100.0kW. Radiation will not be used.

25.4.2 STEP 1: STARTING THE CASE SPECIFICATION TOOL

Run the ***SMARTFIRE*** case specification tool. Once the graphical interface has opened, use the [Region] tab (if not already selected) to show the region editor panel. You will be shown a visual depiction of a default region with no geometric objects defined (See Figure 25-23).

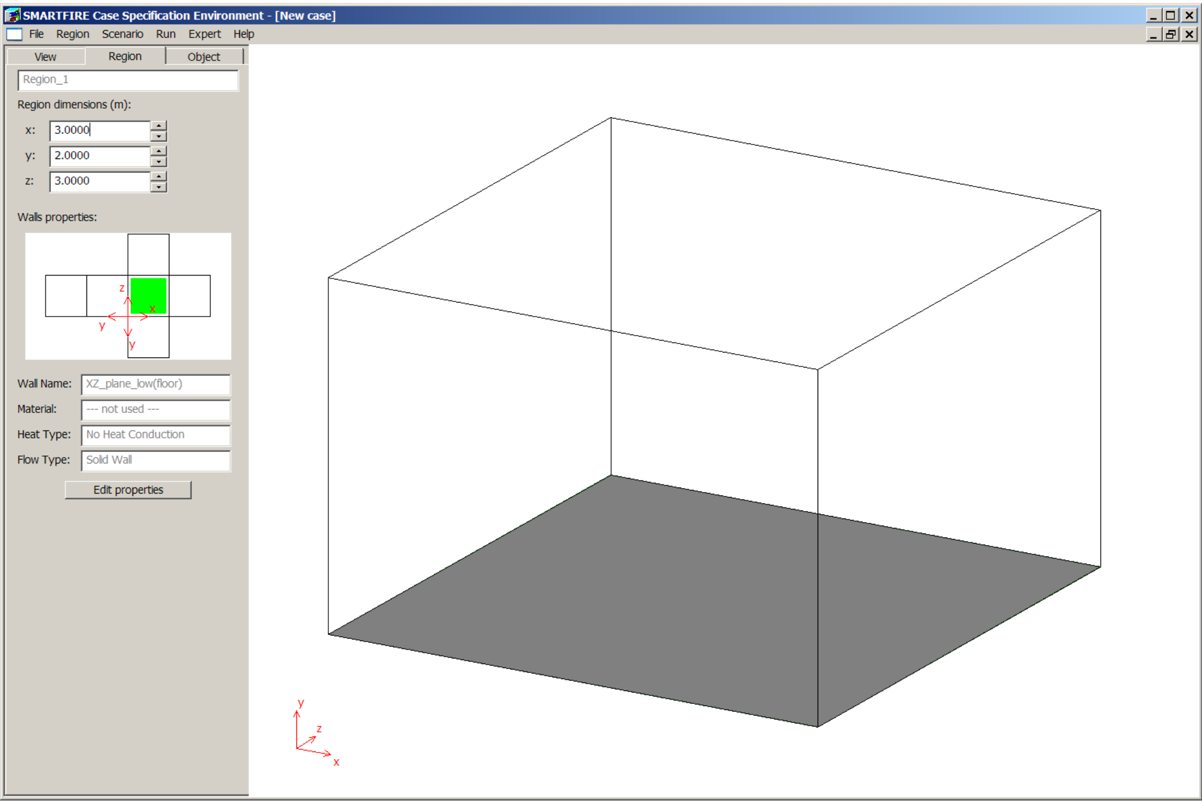

Figure 25-23 Specification tool showing default sized region.

25.4.3 STEP 2: CREATING THE DOMAIN

Enter the dimensions of the region in metres using the three spin boxes. You can select on the spin increment and decrement buttons to change the displayed size by steps of 0.1m or you can double-click the value and type a new one in directly and press the enter button to confirm your input. As specified in the case description above you need the dimensions to be set as follows: x-size = 6.1m, y-size = 2.5m and z-size = 3.0m (See Figure 25-24). You do NOT need to change the wall behaviour for any of the region walls because the default behaviour is acceptable for this case.

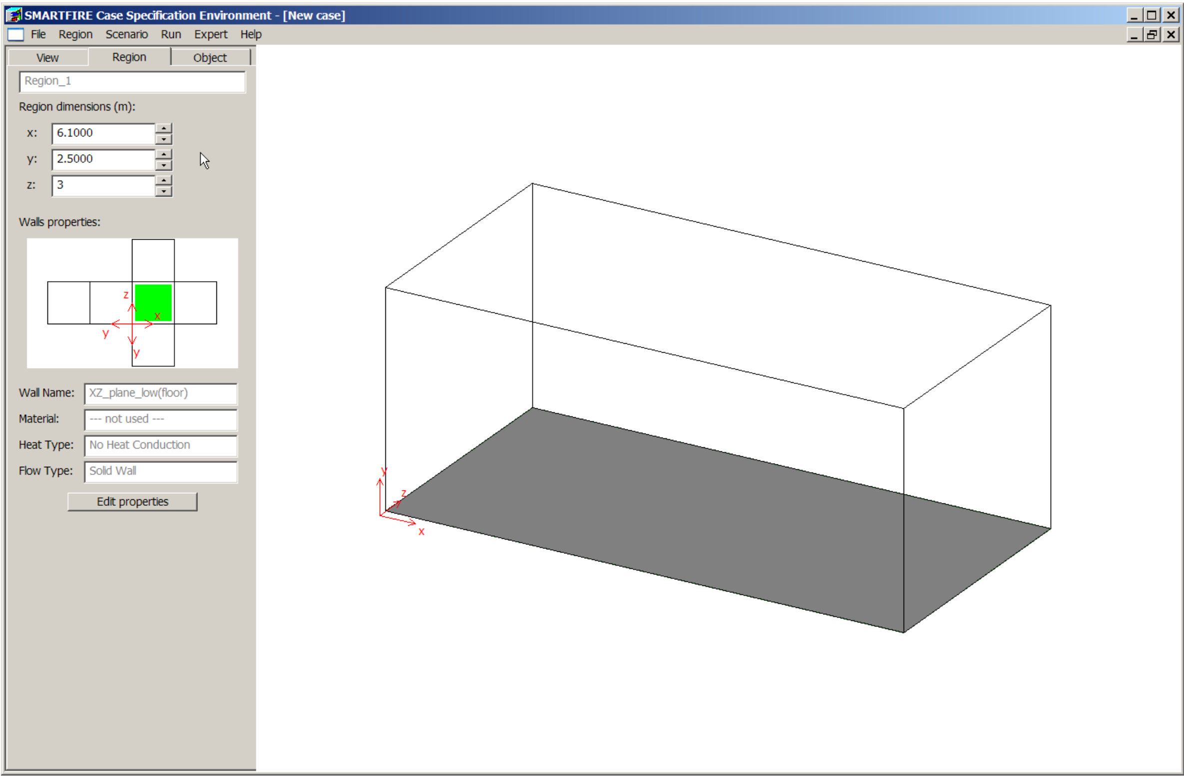

Figure 25-24 Specification tool showing correctly sized region.

25.4.4 STEP 3: CREATING GEOMETRY OBJECTS

Once the region has been sized correctly, you can create all of the geometry objects (fires, obstacles and vents) required for the simulation. In this tutorial you will need to create a fire, two vents (a window and a door), an obstacle (that is the partition wall) and a portal (which makes a doorway through the partition wall).

Select the [Object] tab to display the object editor panel. There are initially no objects defined so the panel only displays a [New Object] button (See Figure 25-25) which you should select to open the new object selection list. The dialogue window presents a list of all of the object types available for inclusion in your simulation. Select the "VENT" object type and press the

[Add] button. The dialogue window will close and a default sized rectangular vent will be added to the region display at a default location (See Figure 25-26). The object editor panel will modify itself to match the currently selected object type (in this case the vent you have just created). The current object will always be displayed using its own typed colour key for the edges and shaded grey faces. Non current objects do not have grey shaded faces.

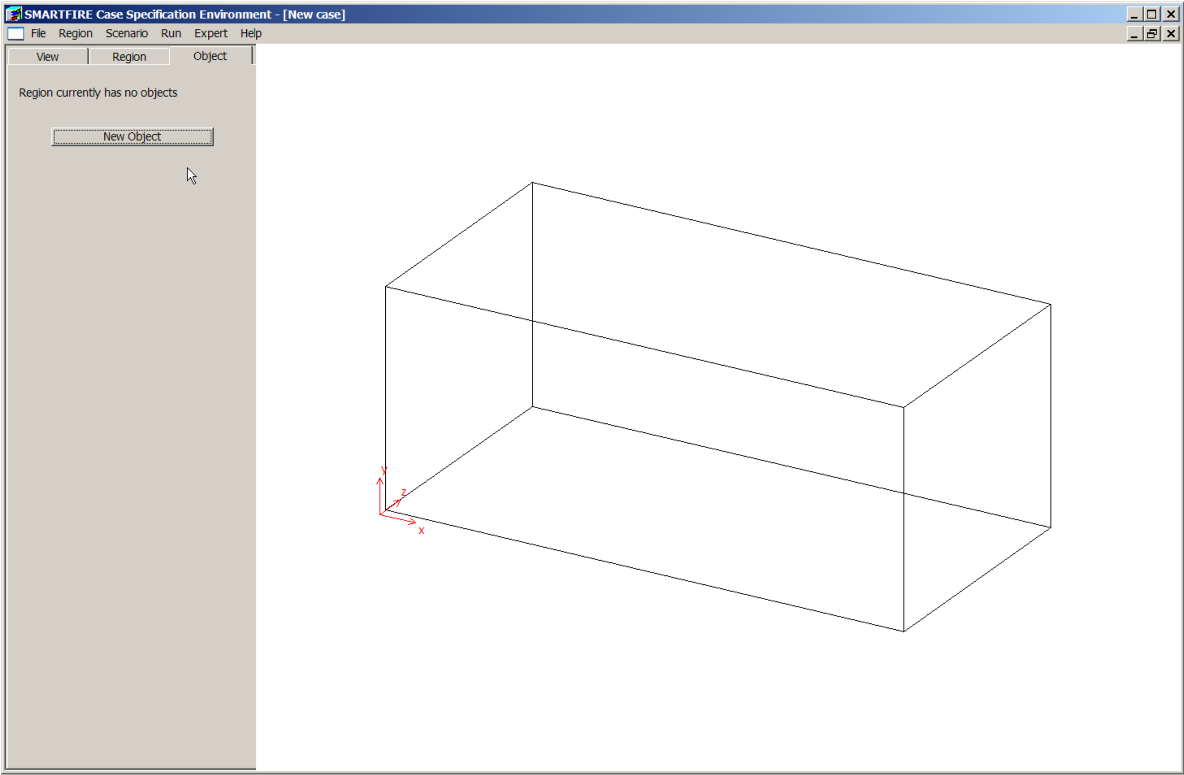

Figure 25-25 Specification tool showing the object panel.

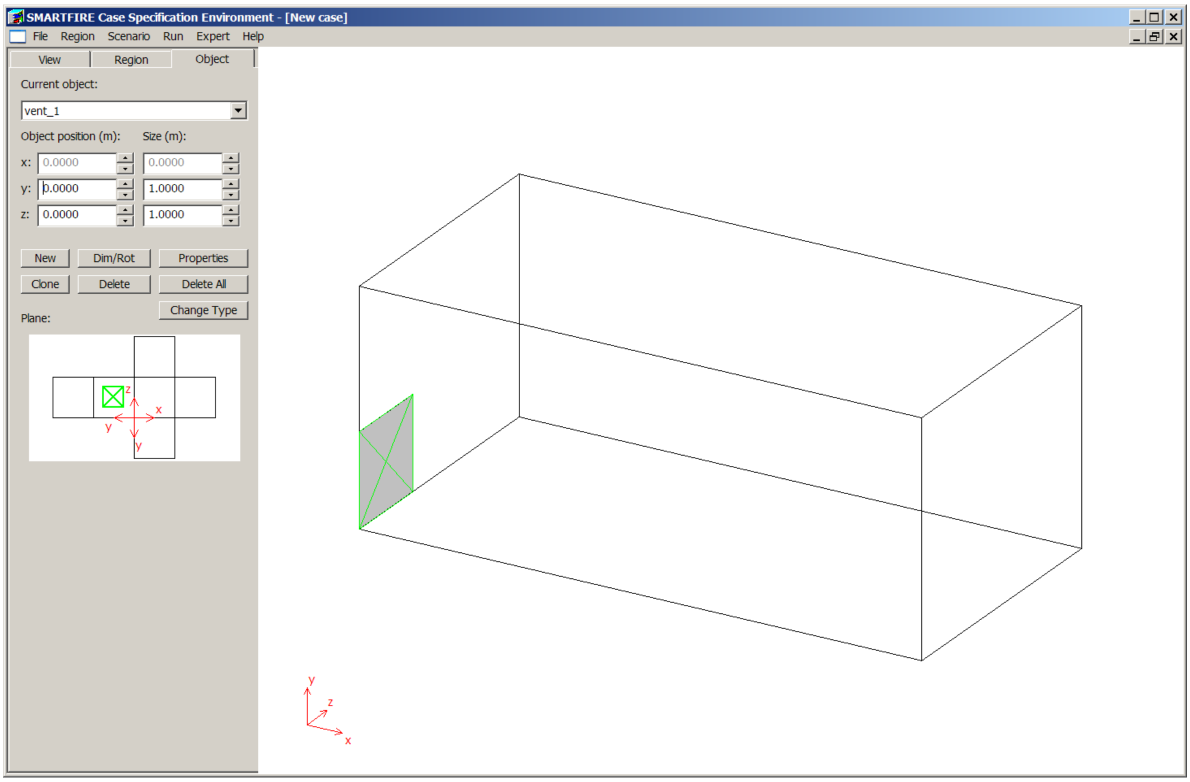

Figure 25-26 Specification tool showing new VENT at the default location.

In this simulation it is required that the vent (doorway) have a width = 1.0m and height = 2.0m and be centrally located on the high-x wall. Use the "*unfolded region*" display area to choose the high-x region face so that the vent appears on the high-x surface of the region.

Enter the size and location of the vent in the spin boxes on the object editor panel. Since this is a constrained 2D object, only relevant spin boxes will be enabled for entering sizes and locations. In the case of the doorway specified in the case description you will need to enter the following values provided that the vent is on the high-x surface: y-size = 2.0m, z-size = 1.0m and y-location = 0.0m and z-location = (3.0m / 2) - (1.0m / 2) = 1.0m. The (door) vent will now be correctly sized and centrally located on the high-x surface of the domain (See Figure 25-27).

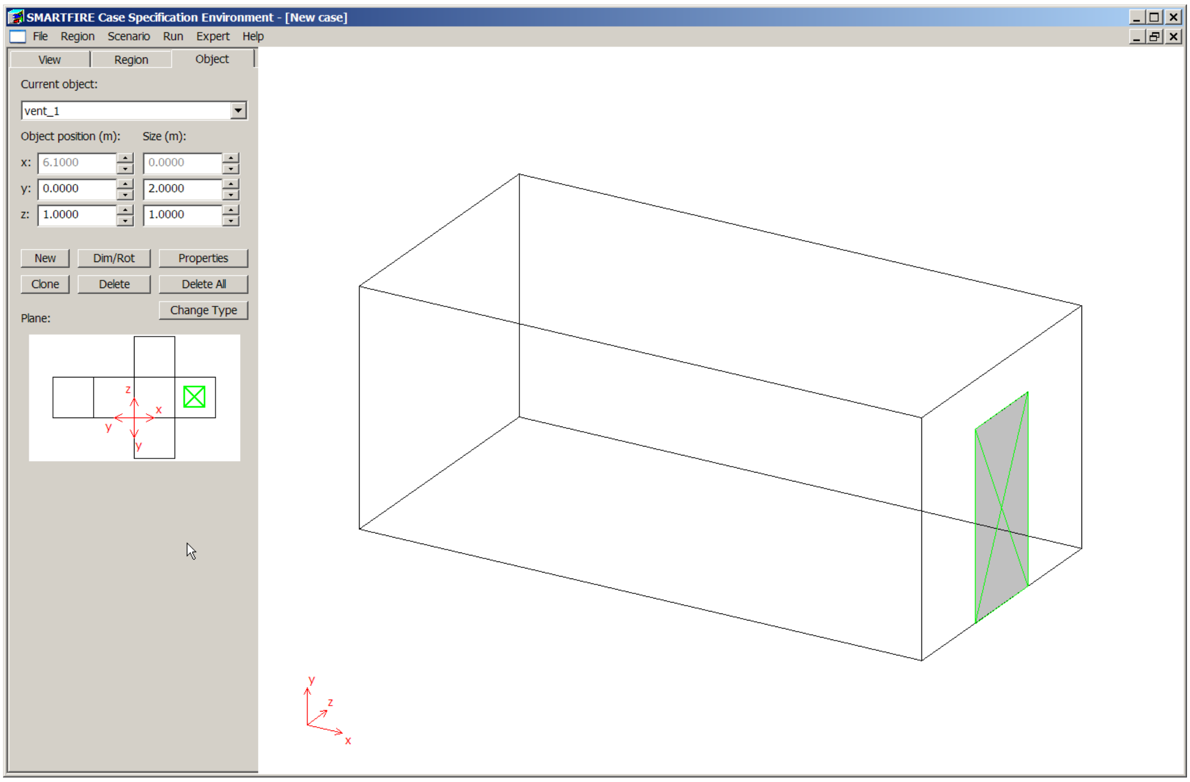

Figure 25-27 Specification tool showing door position and size.

Create another "VENT" object using the [New] button but this time use the "*unfolded region*" display to locate it on the high-z surface of the region. Enter the following sizes and locations for the side window: x-size = 1.0m, y-size = 1.0m, x-location = (3.0m / 2) - (1.0m / 2) = 1.0m and y-location = (2.5m / 2) - (1.0m / 2) = 0.75m. The (window) vent will now be correctly sized and centrally located for the first compartment (See Figure 25-28).

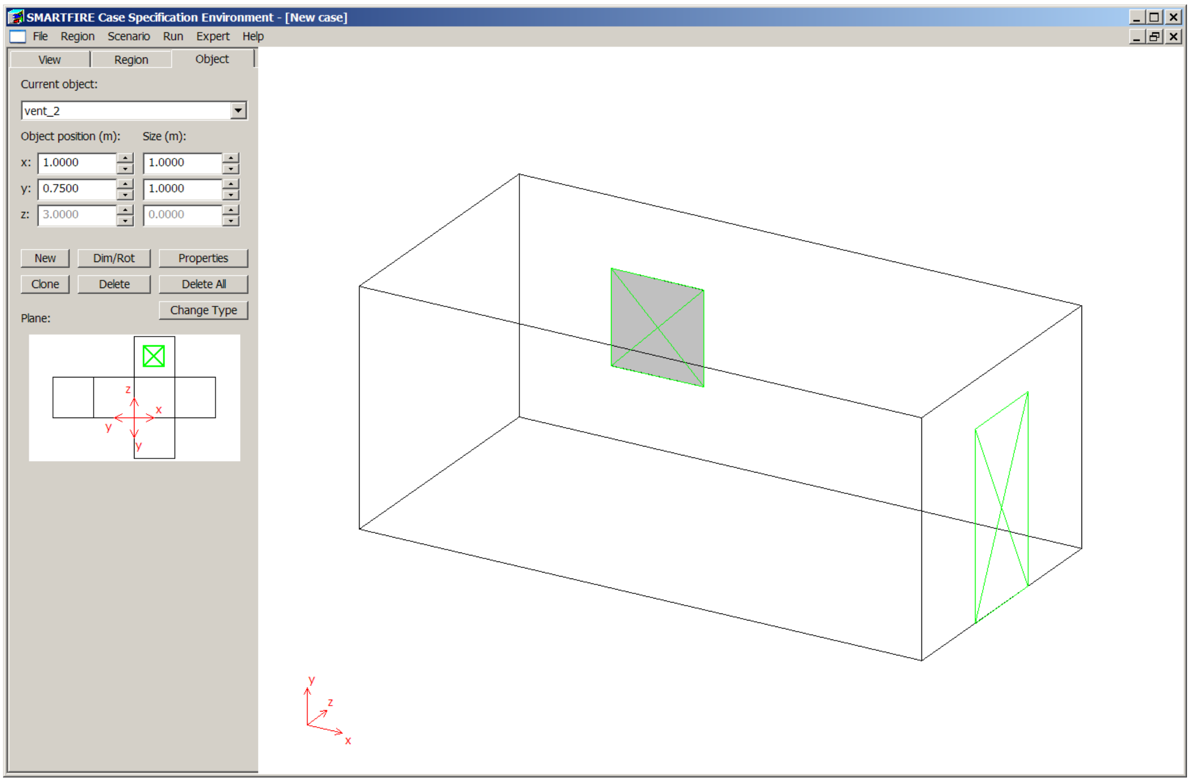

Figure 25-28 Specification tool showing window position and size.

Select the [New] button and choose to add an "OBSTACLE". Enter the following sizes and locations to create the partition between the two compartments: x-size = 0.1m, y-size = 2.5m, z-size = 3.0m dimensions and x-location = 3.0m, y-location = 0.0m, z-location = 0.0m locations (See Figure 25-29).

You may like to note that the OBSTACLE objects that you create will all have the default material type "Wall_Default_Material" which, in this version of ***SMARTFIRE***, is set to concrete. If you require the obstacles to have different material characteristics then you should select the object [Properties] button when the required object is current and choose an appropriate material from the library of material definitions provided. For the purposes of this tutorial you do not need to change the material type.

Select the [New] button and choose to add a "PORTAL". Enter the following sizes and locations to create the second part of a three component partition arch that makes a doorway between the two compartments: x-size = 0.1m, y-size = 2.0m, z-size = 1.0m dimensions and x-location = 3.0m, y-location = 0.0m, z-location = 1.0m locations (See Figure 25-30).

Expert users may wish to use the [Coarse mesh] budget option to allow greater ease of creating a mesh manually. This option makes sure that all objects have at least the minimum number of required internal cells for their correct operation but only puts a single cell in non-critical blocks. The user can then easily use the interactive manual mesh editing tool to add cells as required to each of the under-populated meshing blocks.

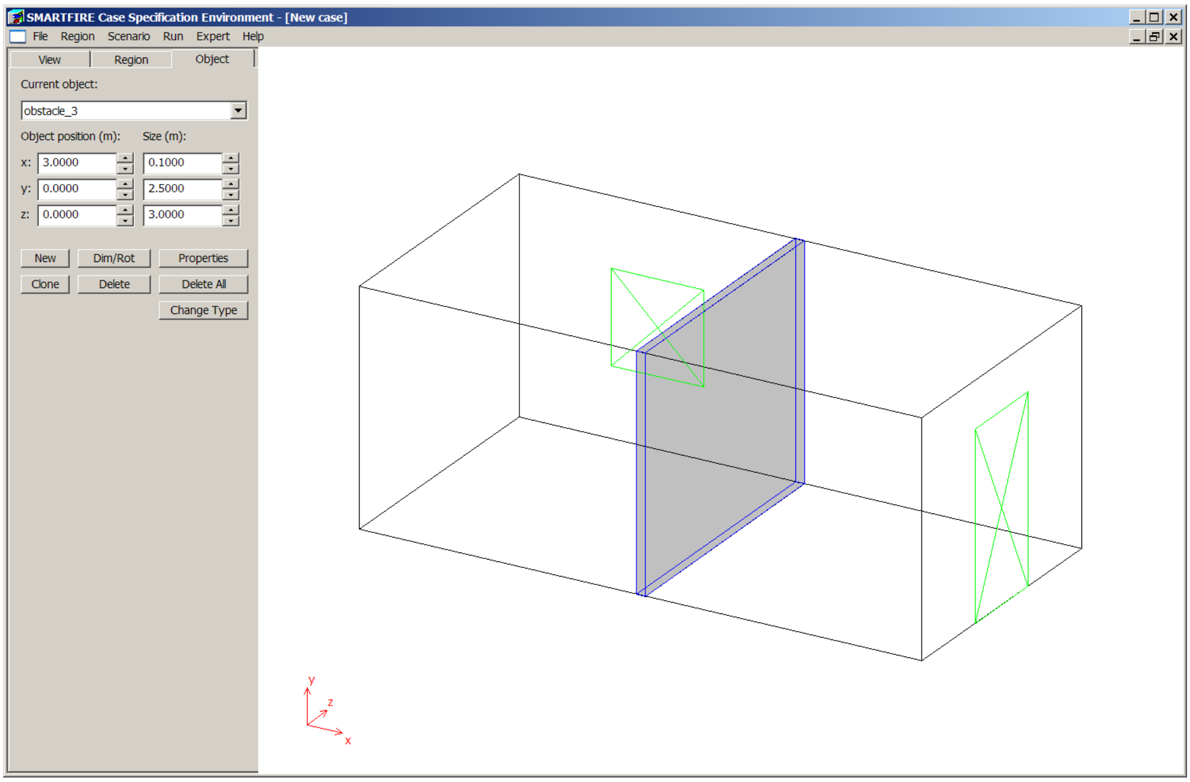

Figure 25-29 Specification tool showing the partition.

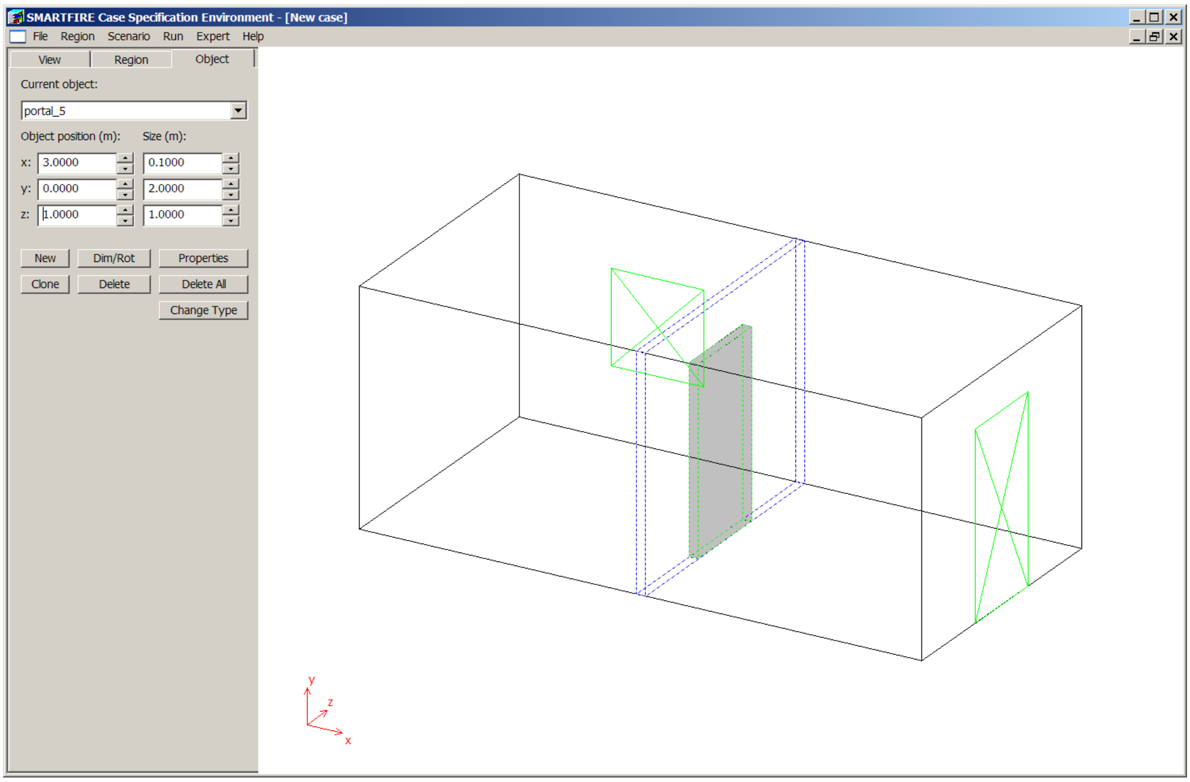

Figure 25-30 Specification tool showing a portal connecting the two compartments through the partition.

The final object that needs to be created is the fire source. Select the button labelled [New] in order to choose another object type to add to the region. This time select the "SIMPLE_FIRE" type from the list and press the [Add] button. A default sized cube will placed at the low co-ordinate (0.0, 0.0, 0.0) location of the region and will be made current (grey faces). Enter the sizes of this fire using the spin boxes. In this case the dimensions are x-size = 0.5m, y-size = 1.0m and z-size = 0.5m. The locations are x-location = (3.0m / 2) - (0.5m / 2) = 1.0m, y-location = 0.0m and z-location = (3.0m / 2) - (0.5m / 2) = 1.0m. Enter these values in the appropriate spin boxes and the fire will be centrally located in the first compartment and correctly sized (See Figure 25-31).

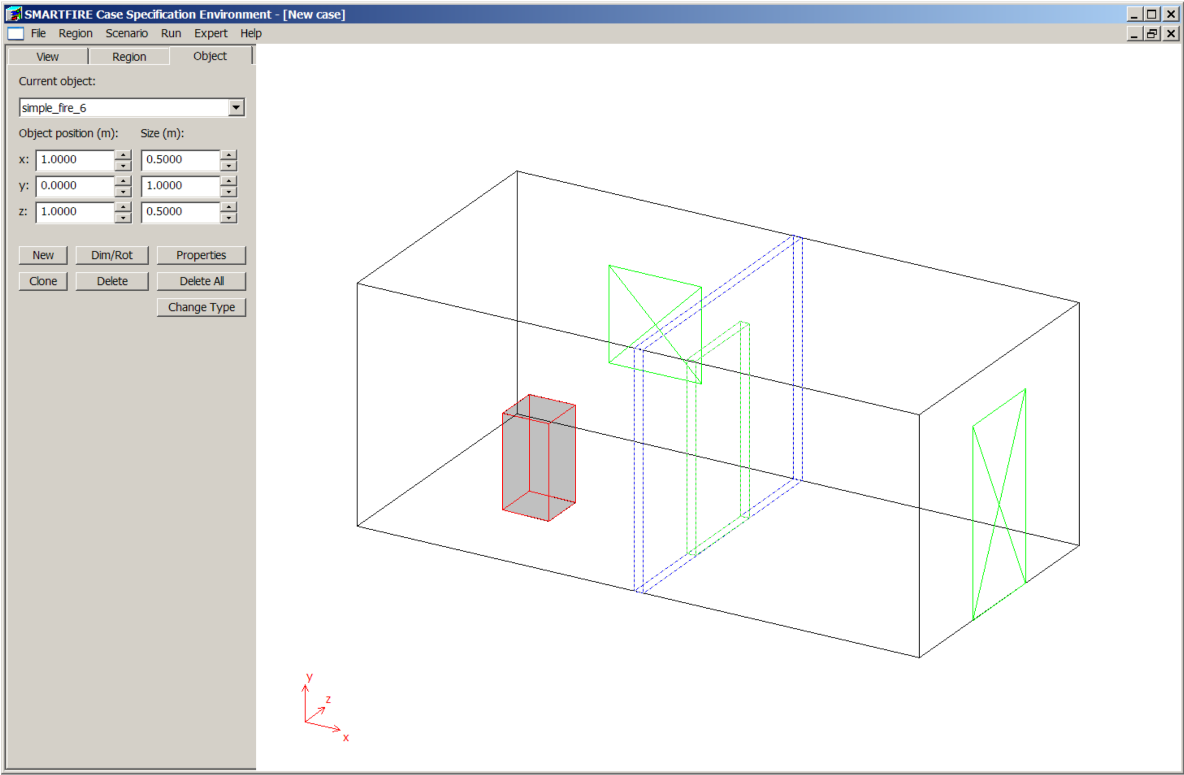

Figure 25-31 Specification tool showing fire location and size.

The fire will have a default fire output curve specified (constant 50kW fire) which is not appropriate to this case specification. Select the button labelled [Properties] to access the fire properties window.

The fire properties window allows you to choose a fire behaviour curve from a number of given or parametric heat release curves. In the case description the fire is required to be a constant heat output value of 100.0kW. Select the [Constant] power curve type radio button to enable the constant curve and enter 100.0 into the [A] value spin box (See Figure 25-32). All of the other spin box values will be disabled for constant power fires. There is no need to change the fire start and end times because constant fires are assumed to start at t=0.0 seconds and continue indefinitely. Select the [OK] button to close the dialogue.

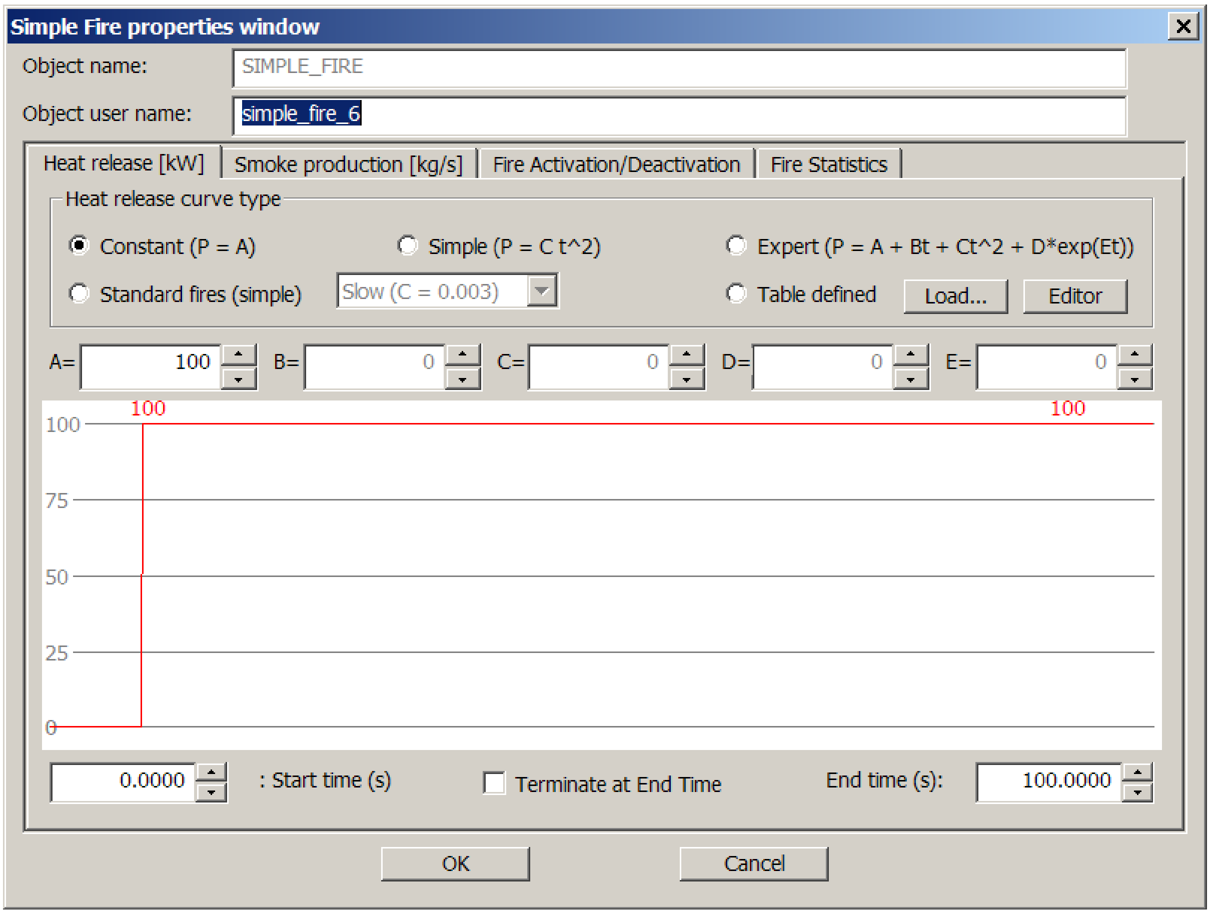

Figure 25-32 Fire properties window showing constant fire of 100.0kW.

25.4.5 STEP 4: DEFINING THE PROBLEM TYPE

Select the [Scenario] menu from the main menu bar and choose the [Problem Type] item. This will open a configuration window that allows the simulation controls to be modified (See Figure 25-33). It is estimated that this case will reach near steady state after 30 minutes or so BUT indicative results can be obtained after 3 minutes. In order to perform a reasonably short simulation it will be necessary to change the simulation time characteristics. An acceptable simulation can be performed using 18 time steps of 10 seconds per time step and 50 sweeps per time step. Enter the following values to configure the simulation characteristics: Time step size = 10.0s, Number of time steps = 18 and Number of sweeps per time step = 50. This will give (18 x 10) = 180 seconds (i.e. 3 minutes) of simulated time.

Radiation should not be enabled so make sure that the check box for the Radiation activation is empty (i.e. not ticked or crossed). None of the other parameters need to be changed from their default values but the user is advised to glance at the other global control items that can be configured in this window.

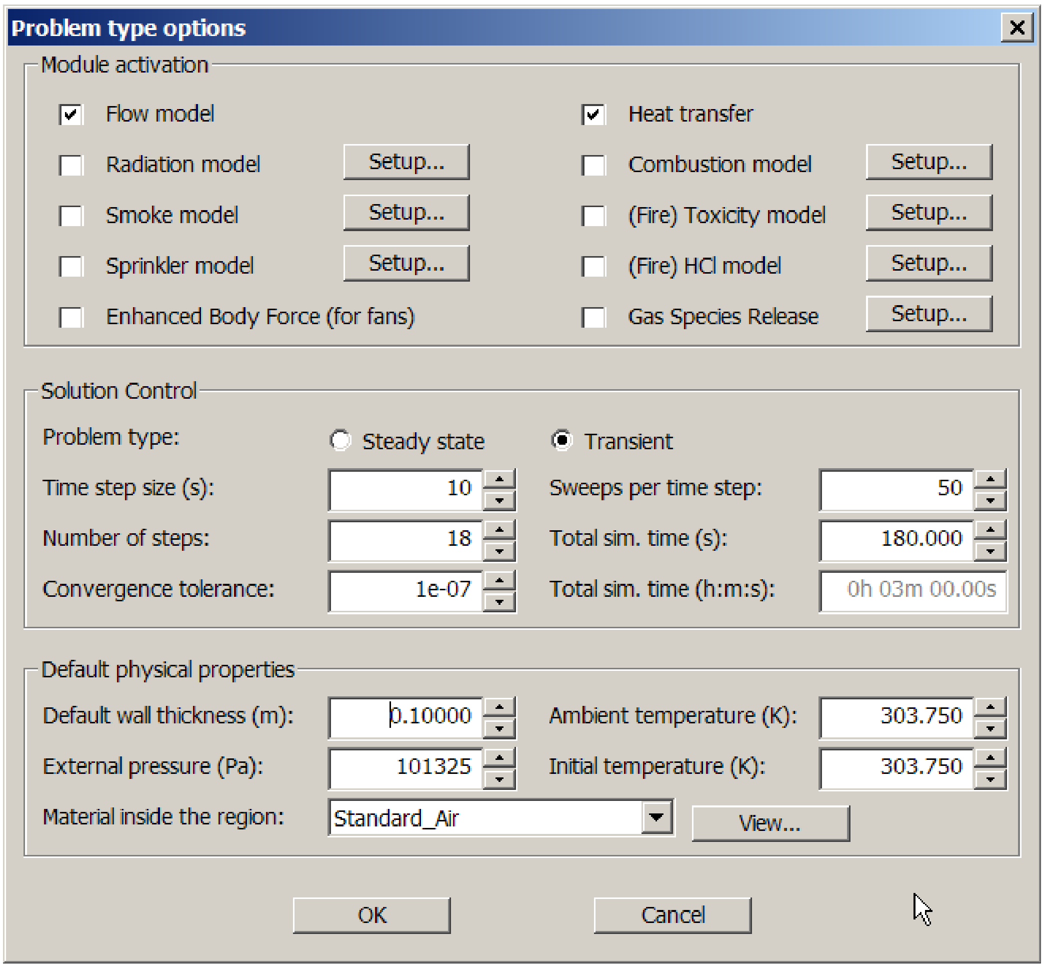

Figure 25-33 Problem type dialogue.

25.4.6 STEP 5: SAVING SPECIFICATION TO A NEW CASE DIRECTORY

Now that the simulation has been fully specified it is necessary to save the geometry and physics settings to a directory where all other case-related simulation files will be created, accessed and stored. Select the main menu [File] option and the [Save As] item. This opens a file browser in the work directory. Enter a unique name for this case (See Figure 25-34).

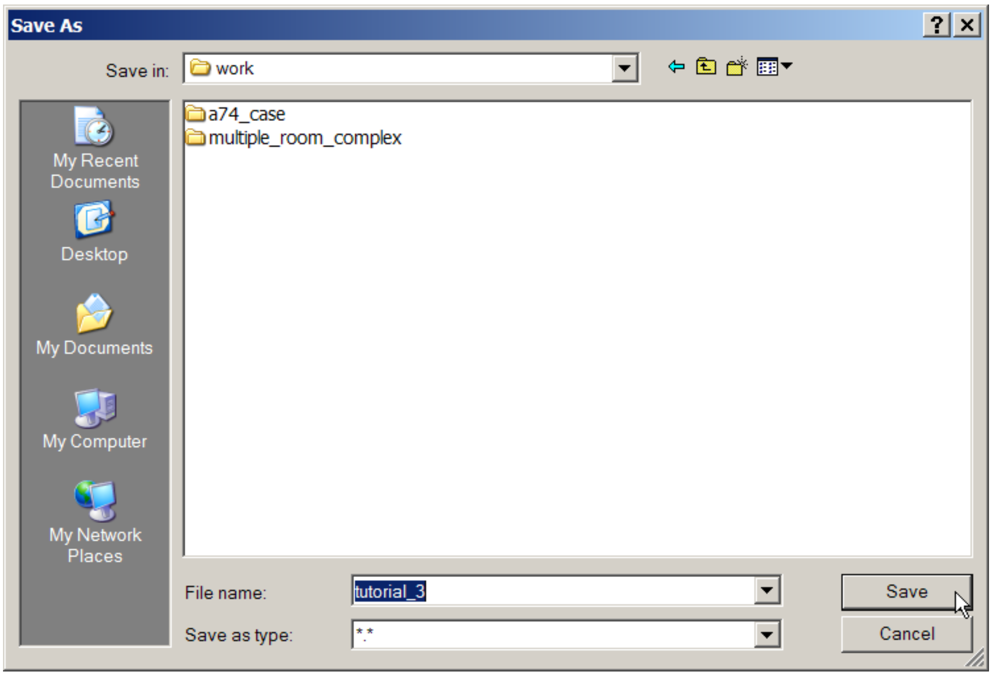

Figure 25-34 File [Save As] dialogue being used to give a name to the case.

It is recommended that you choose a representative name that you can use to easily identify this case again in the future. For example you could enter the name "Tutorial_3" and then select the [Save] button. This will create a directory (folder) called Tutorial_3 and will save the specification file "Tutorial_3.smf" into the directory. You should not create case directories yourself since ***SMARTFIRE*** expects to do this for you when you save a new case.

25.4.7 STEP 6: FINISH SPECIFICATION AND START CFD ENGINE

You are now ready to create a mesh specification, a specification script file and to simulate this case in the CFD engine. The steps to achieve this are described here in succinct form:

- **Use the main menu [Run] and [Create Mesh] to run the automated meshing system.**
- **Choose a cell budget (as NX, NY and NZ) for this case (it is recommended that a total cell budget of 20000 or so cells will give a reasonable quality mesh for this scenario).**
- **[Accept] the mesh to create the geometry file and specification script - this will also exit from the automated meshing tool.**
- **Start the CFD engine by selecting the main menu [Run] and [Run CFD Engine].**
- **Wait for the CFD Engine to initialise.**
- **Start the CFD Engine using the [Run] button.**

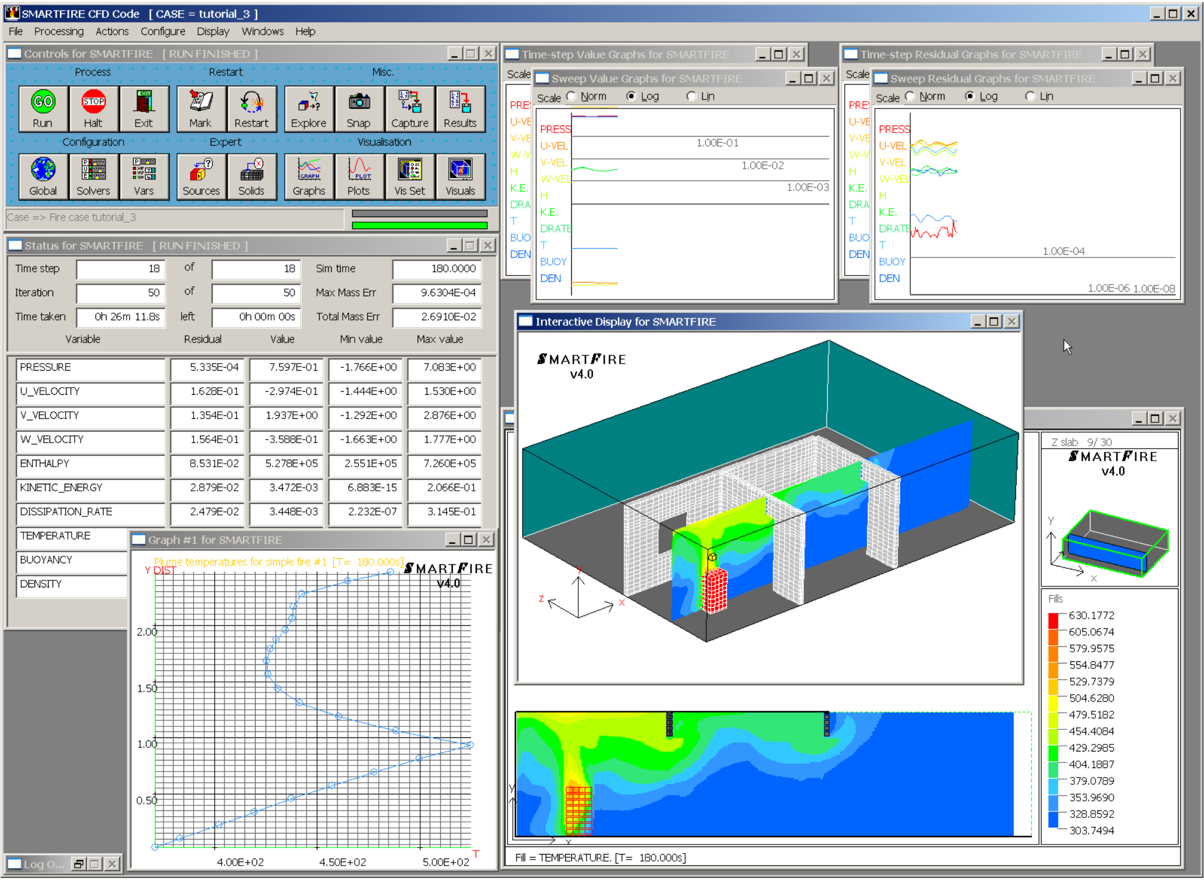

Figure 25-35 CFD engine at end of full simulation (rearranged windows).

At any stage of the processing you may stop the CFD engine, using the [Halt] button, in order to change the visualisation view, to explore the data, to define graphs or to modify the control parameters BUT you must wait for the current sweep to end (upper green progress bar must have reached the right hand edge in the Control window).

The final image shows the end of the configured simulation (See Figure 25-35). You should notice that the User Interface windows have been reorganised slightly to present this (slightly elongated) case more clearly.

Stop the CFD code and press the Control panel [Exit] button when you want to finish.

This is the end of tutorial 3.

25.5 *TUTORIAL 4*

25.5.1 OVERVIEW

This tutorial takes the user through the specification of a moderately complex multiple room geometry using the ***SMARTFIRE*** case specification tool, meshing the case using the automated interactive meshing tool and simulating the case using the ***SMARTFIRE*** CFD Engine. The simulation case represents one corner of a larger building where the fire analysis is to be conducted. This study is based around a large fire in the main corridor and investigates the impact this has on the temperatures in the adjacent rooms. The plan view of the geometry is as follows:

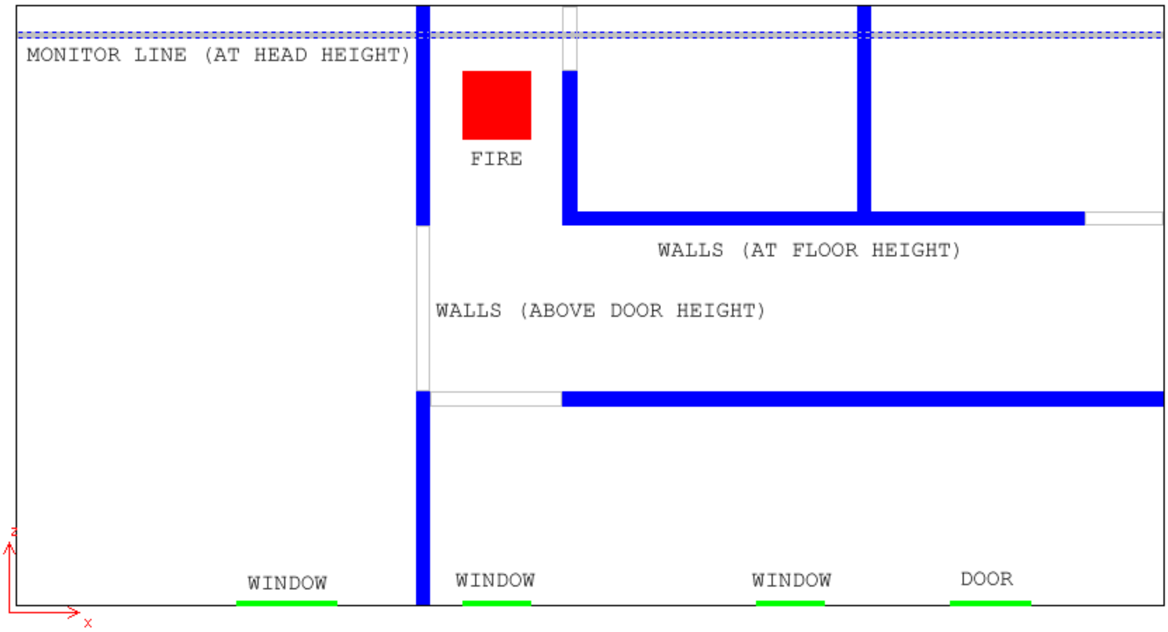

Figure 25-36 Plan view drawing showing the room layout geometry.

Certain assumptions have been made to simplify the specification as follows:

- The rooms are all assumed to be open plan, and all of the doorways and windows are fully open for all of the simulation.
- The doors on the end of the corridor are assumed closed throughout the simulation and that the fire will not affect their structural integrity. There are actually windows present on the large outer wall of the largest room but these are assumed closed (and are hence not modelled) during the period of the simulation. It should be noted that the windows could be made open, but this would require an additional extended region and hence require additional simulation time.
- Small air leaks around closed doors or windows are ignored and the heat transfer through the walls is considered to be negligible compared to heat transfer due to other processes, in the duration of the simulation. Again, these are merely simplifying

assumptions for the purposes of this example. They can be accommodated within a SMARTFIRE simulation but are not used in this demonstration.

- The corridor is assumed to have no combustible materials, which can support fire spread or secondary ignition.

The geometry of this case requires a large number of OBSTACLE objects to make the internal walls. It is assumed that the user in familiar with the construction, correct sizing and positioning of such objects - from the previous tutorials.

The fire is assumed to be a cleaning trolley fire at one end of the corridor. The fire reaches a peak heat release rate of 2MW in two minutes. Since the fire is in the corridor and will prevent conventional means of egress from the building, we are actually interested in determining the time to reach a critical temperature (at about head height) in the neighbouring rooms where there are no exterior doorways to use for exit.

25.5.2 STEP 1: CREATING THE GEOMETRY AND SCENARIO

Run the ***SMARTFIRE*** case specification tool. Once the graphical user interface has opened, select the [Region] menu panel and set the size of the geometry region as X=17.2m, Y=4.0m and Z=9.0m. Having correctly sized the simulation region, it is sensible to name the case by selecting the [File] and [Save As] option. This allows the creation of the ***SMARTFIRE*** work directory for this case. All of the model files and result files, for the current case, will reside in this folder. For example if the case is named "t_4" then the file "t_4.smf" will be saved in the "smartfire\work\t_4" directory. This folder will contain all of the geometric and physical properties for the specification of the case and, later it will contain all of the simulation and result files for this scenario.

The simulation scenario uses Heat, Flow and the Radiation model with the Six-Flux model activated. The Default Wall Emissivity is set to 0.8. The Combustion, Smoke and Enhanced Body Force models are NOT activated. The simulation is run in Transient mode with a Time Step Size of 2.0 seconds with 50 Sweeps per time step. The Number of time steps is set to 150 (i.e. simulating 300 seconds or 5 minutes). The Convergence tolerance is used as the default 0.001. The physical properties are set as follows. The Default wall thickness is 0.2m, the Ambient temperature is set to 288.75K, the External pressure is 101325.0 Pa and the Material inside the region is "Standard_Air".

Select the [Scenario] option from the main menu bar and select [Problem type] to access the Problem type for this scenario. The Problem type options window should appear as follows:

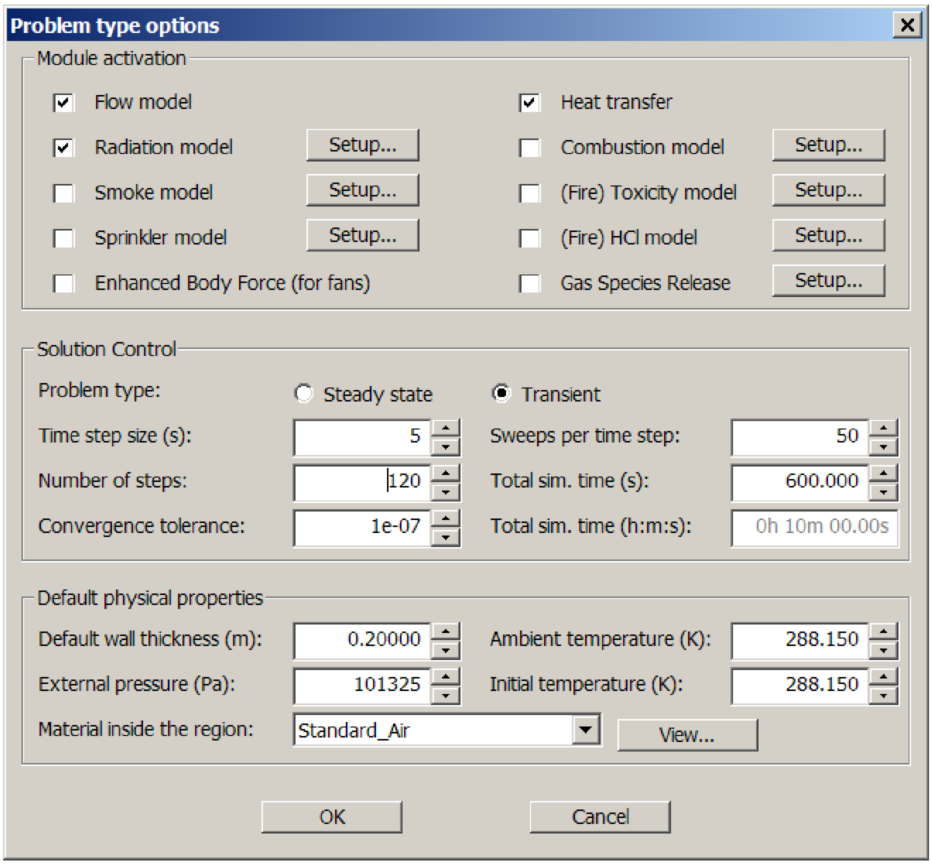

Figure 25-37 Problem type options window.

In order to create the objects that make the walls, windows and the fire for the simulation case, select the [Object] notebook tab. This will allow you to add objects to the current region or to change the properties of any objects already created. You will need to create, accurately position and correctly size the following list of objects by selecting [New]. The objects that need to be created are as listed below. It should be noted that some considerable time could be saved when creating a large number of objects by using the [Clone] feature. It is possible to create the first Wall Partition and set the correct material type, and then to clone the object and move and resize the "cloned" partition to the position and size of the next Wall Partition. This saves the time associated with setting the material type for the "cloned" object and it is often the case that one or more of the position and/or size parameters will be the same as the previous object. Similarly the Windows can be cloned and then suitably sized and positioned further along the wall. When an object is cloned the clone is created in the same position and with the same size as the "parent" object. Both the cloned and parent objects will be displayed with dotted edges (in wire frame view mode) to indicate that they are "overlapping" objects.

The following objects define the geometry of the multiple room simulation scenario.

Window (in the large room):
VENT at (x=3.3, y=1.0) on the low-z wall. Size is (dx=1.5, dy=2.0).

Window (in the long thin room):
VENT at (x=6.7, y=1.0) on the low-z wall. Size is (dx=1.0, dy=2.0).

Window (in the long thin room):

VENT at (x=11.1, y=1.0) on the low-z wall. Size is (dx=1.0, dy=2.0).

Doorway (in the long thin room):
VENT at (x=14.0, y=0.0) on the low-z wall. Size is (dx=1.2, dy=2.5).

Partition Wall (wall of large room):
OBSTACLE at (x=6.0, y=0.0, z=0.0). Size is (dx=0.2, dy=4.0, dz=9.0). Material=Brick.

Doorway (in wall of large room):
PORTAL at (x=6.0, y=0.0, z=3.2). Size is (dx=0.2, dy = 2.5, dz=2.5).

Partition Wall (wall of long thin room):
OBSTACLE at (x=6.2, y=0.0, z=3.0). Size is (dx=11.0, dy=4.0, dz=0.2). Material=Brick.

Doorway (in wall of large room):
PORTAL at (x=6.2, y=0.0, z=3.0). Size is (dx=2.0, dy = 2.5, dz=0.2).

Partition Wall (wall of little room nearest the fire):
OBSTACLE at (x=8.2, y=0.0, z=5.9). Size is (dx=0.2, dy=4.0, dz=3.1). Material=Brick.

Doorway (doorway of little room nearest the fire):
PORTAL at (x = 8.2, y=0.0, z=8.0). Size is (dx=0.2, dy=2.5, dz=1.0)

Partition Wall (corridor edge wall for both little rooms):
OBSTACLE at (x=8.2, y=0.0, z=5.7). Size is (dx=9.0, dy=4.0, dz=0.2). Material=Brick.
Doorway (farthest from fire)
PORTAL at (x =16.0, y = 0.0, z=5.7). Size is (dx=1.2, dy=2.5, dz=0.2).

Partition Wall (wall between the little rooms):
OBSTACLE at (x=12.6, y=0.0, z=5.9). Size is (dx=0.2, dy=4.0, dz=3.1). Material=Brick.

Fire :
SIMPLE FIRE at (x=6.7, y=0.0, z=7.0). Size is (dx=1.0, dy=1.0, dz=1.0). Expert Heat Release curve with A=150, B=0.0, C=0.1285, D=0.0 and E=0.0 with Start_time=0.0s and End_time=120.0s. Shows maximum heat release as 2000kW at T=120s.

Monitor Graph :
MONITOR LINE at (x=0.0, y=1.8, z=8.5). Size is (dx=17.2, dy=0.2, dz=0.2). Direction=X-Direction with X axis variable=X_COORD and Y axis variable=TEMPERATURE.

The OBSTACLE objects are defined as follows:

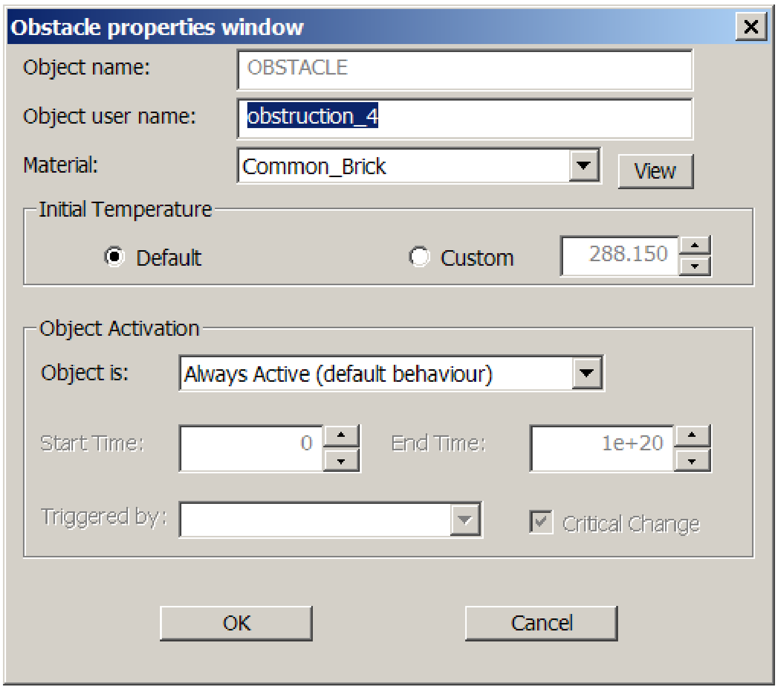

Figure 25-38 Obstacle properties used in the simulation scenario.

The SIMPLE FIRE object is defined as follows:

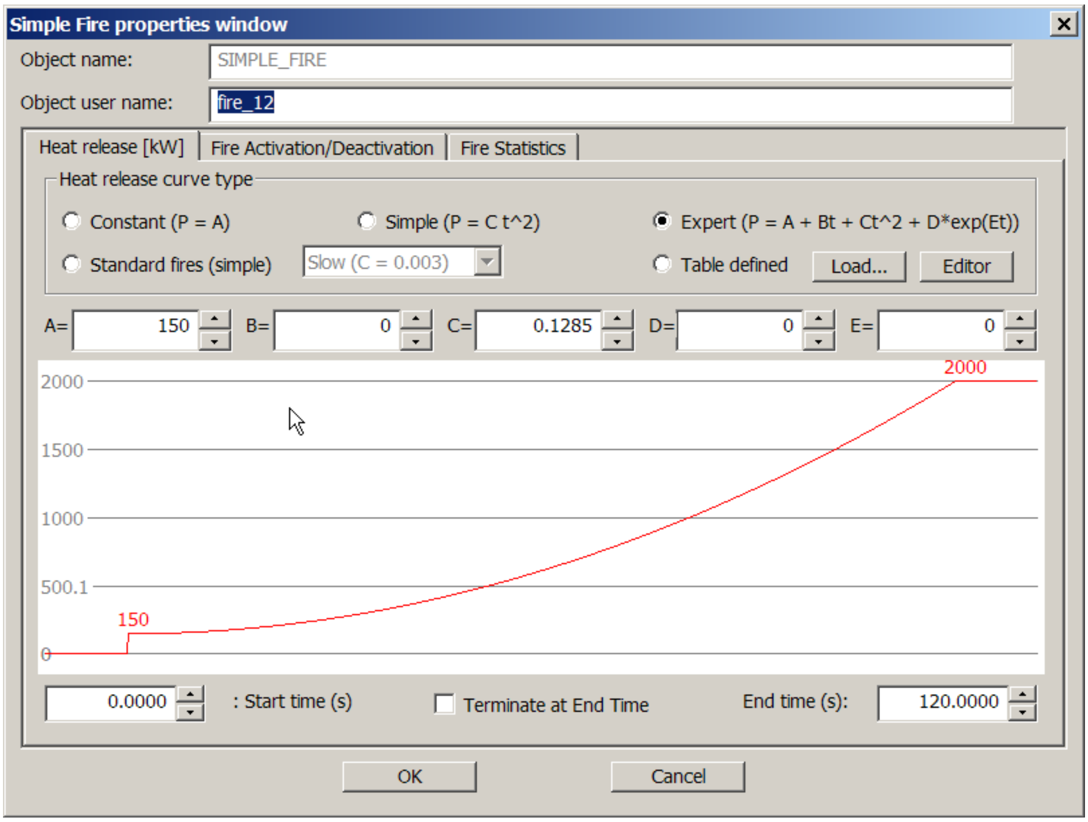

Figure 25-39 Simple Fire properties window showing the fire settings.

The MONITOR LINE object is defined as follows:

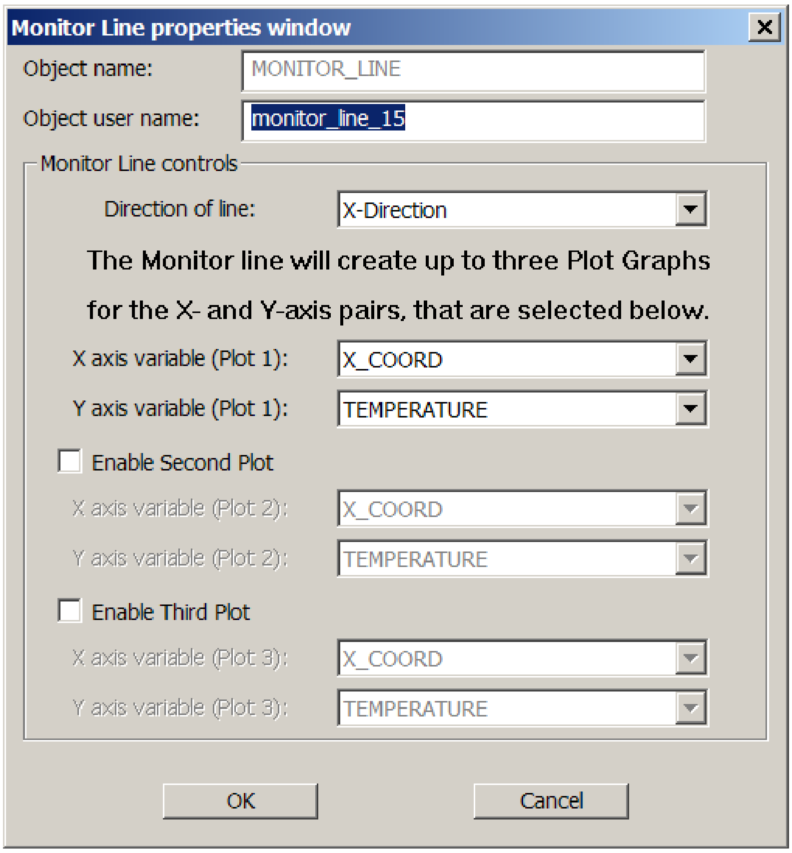

Figure 25-40 Monitor Line properties window showing the settings.

Once all of the geometry objects have been created, positioned and sized correctly the geometry displays should be as follows:

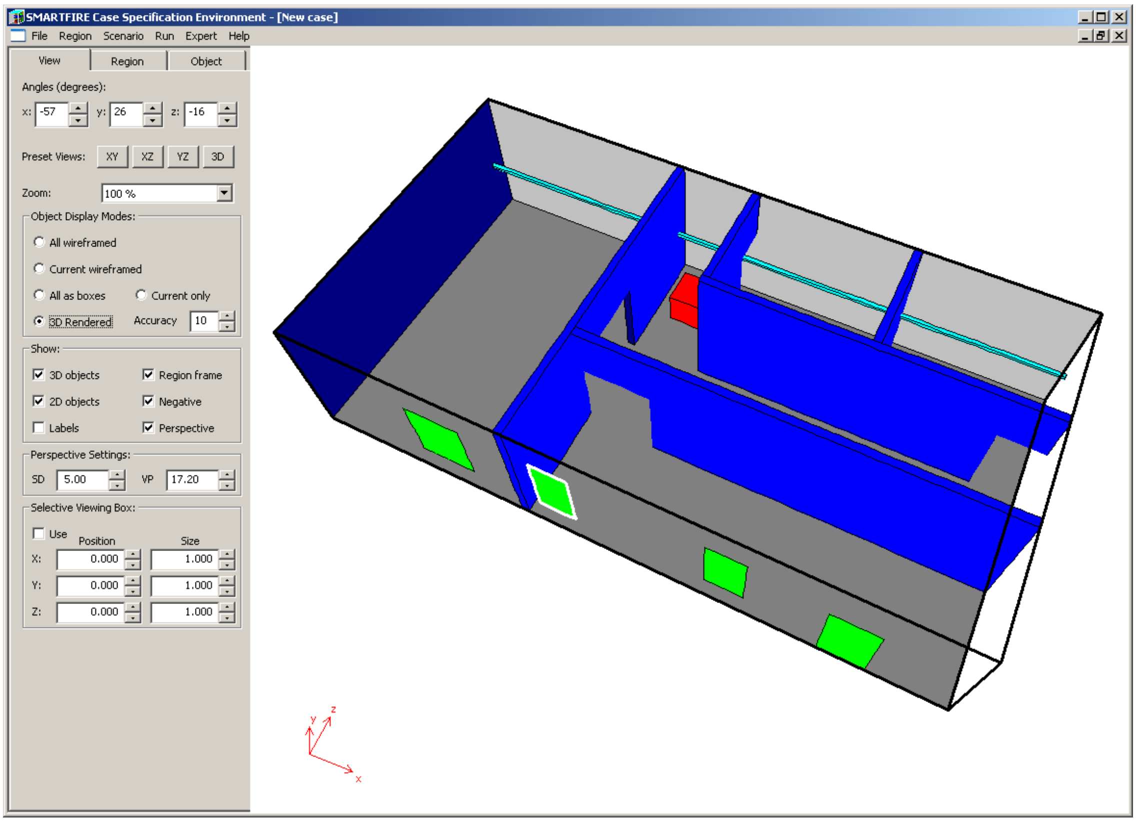

Figure 25-41 Rendered Perspective 3D geometry showing windows, walls, fire and monitor line.

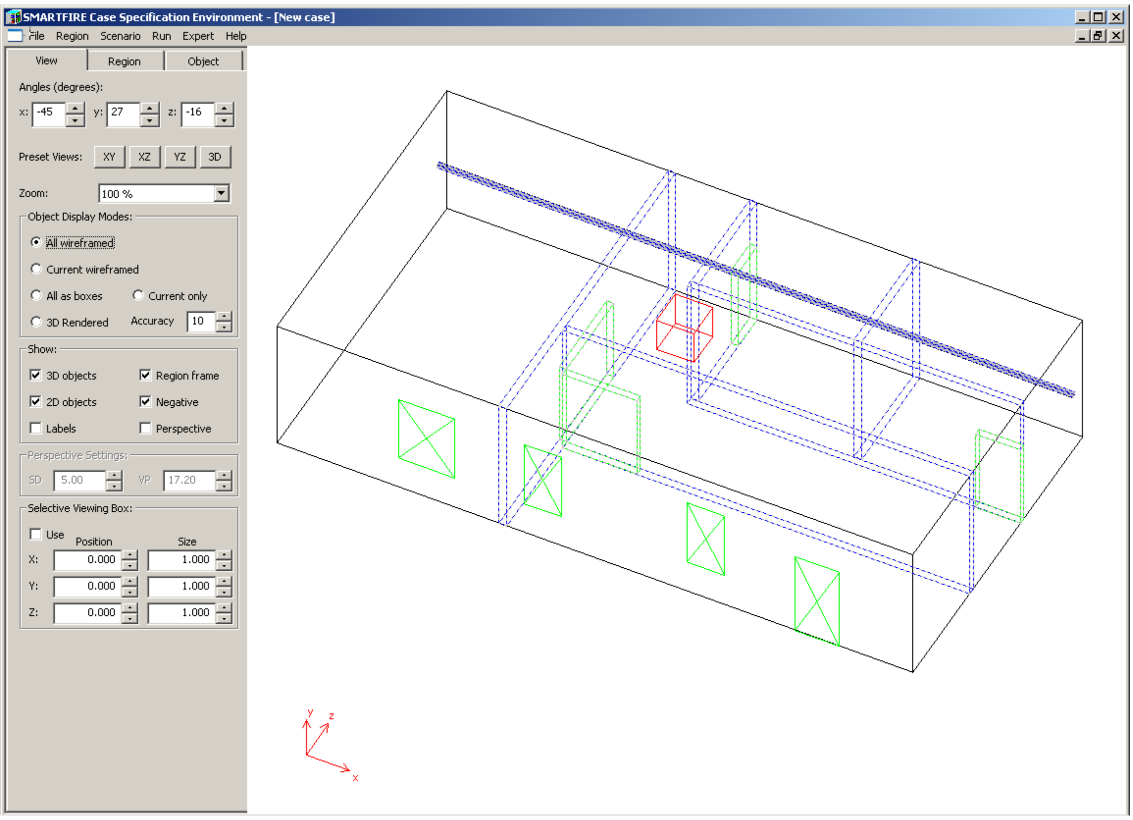

Figure 25-42 Geometry shown in wire frame view mode.

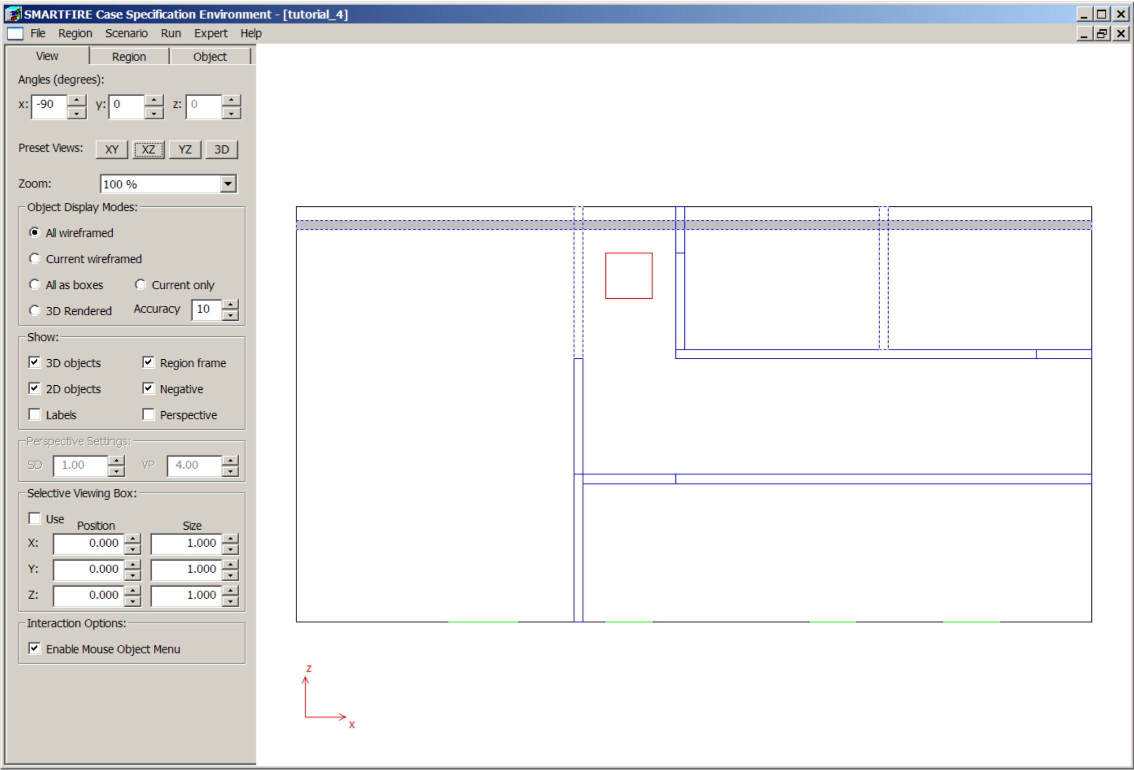

Figure 25-43 Plan view (XZ) display of the geometry.

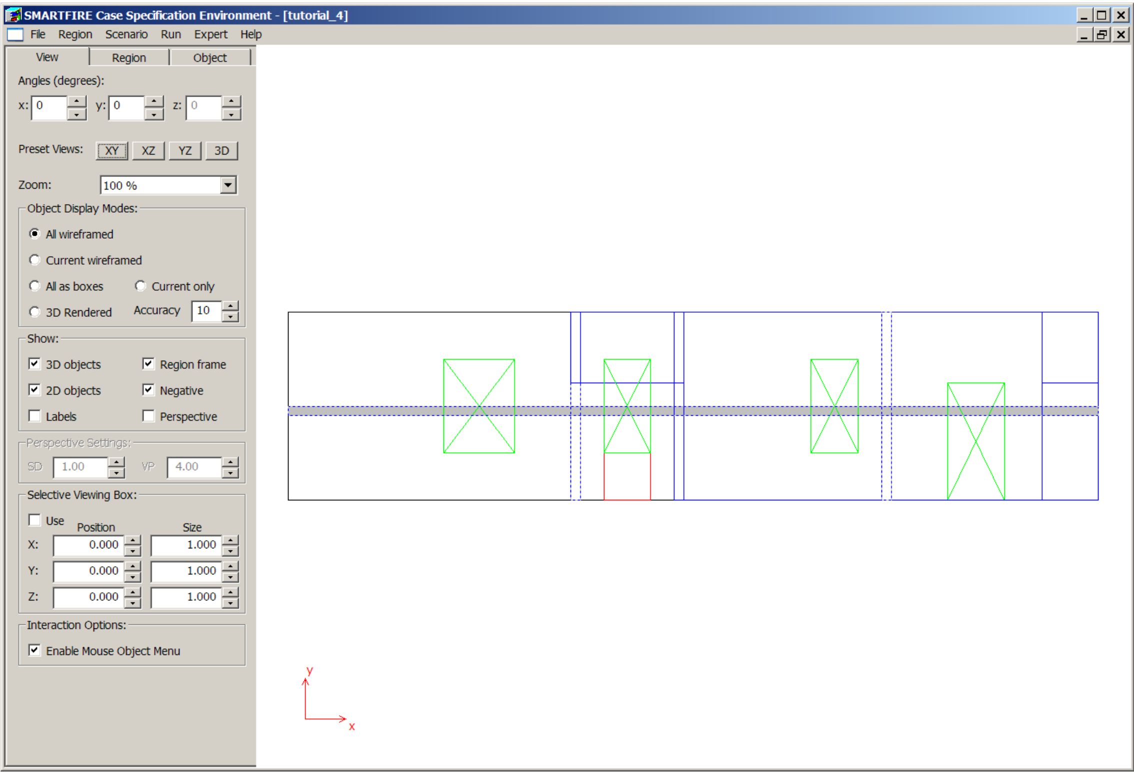

Figure 25-44 Side view (XY) display showing the geometry.

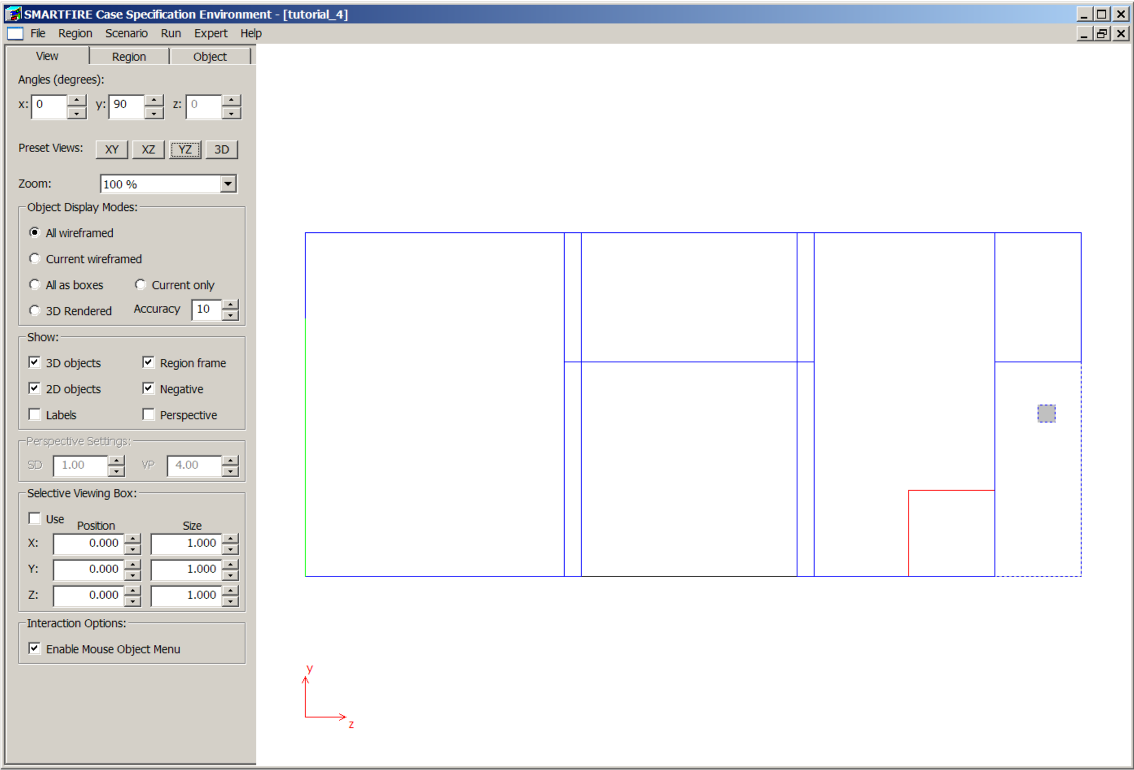

Figure 25-45 End view (YZ) display showing the geometry.

There is one last issue to handle before the geometry can be meshed. The default meshing controls assume that every simulation case should have a sufficiently refined mesh to allow reasonable results to be obtained. This is a reasonable assumption for normal usage, but for the purposes of this tutorial it is recommended that a few of the meshing rules are deactivated to allow a courser mesh to be created. Access to the meshing controls and parameters can be found in the main menu under [Scenario] and [Meshing Controls]. The Meshing Control Parameters window appears as follows:

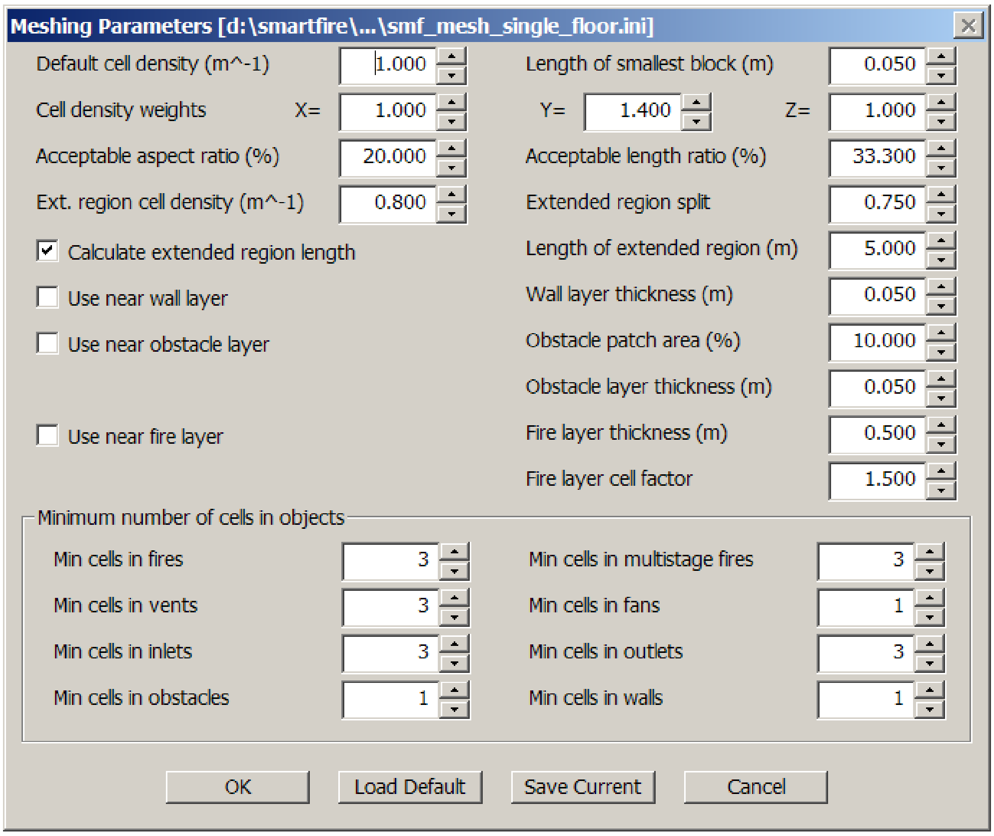

Figure 25-46 Meshing Control Parameters window.

In order to allow the meshing system to create a more even course mesh it is advisable to disable the creation of thin layers around objects. The check boxes that must be unchecked are the [Use near wall layer], the [Use near obstacle layer] and the [Use near fire layer]. All of the other options can be left to the default settings. Do not press the [Save Current] button as this will save the current meshing parameters into a support file to be used whenever the Smartfire Environment is run. Simply select [OK] to proceed with the modified control settings.

25.5.3 STEP 2: CREATING A MESH FOR THE SIMULATION

Once you are satisfied with the geometry and physics handling of the case, you need to run the mesh creation system. To run the automated meshing tool from the main menu, choose the [Run] option and the select [Create Mesh]. This will open the mesh creation tool user interface. The automated meshing tool will first determine if the current geometry already has an existing mesh previously created. If one is available then the user will be given the option to load the existing mesh or to create a new one. It is assumed that no existing mesh is available or that the user will create a new mesh.

The meshing tool will check for any geometry or modelling issues that might need to be resolved. If there are any problems then the system will ask the user to select an appropriate resolving action. For example the system WILL detect overlapping objects in the geometry that has been specified above (the walls and the monitor line object overlap since the monitor line extends from end to end of the region passing through some of the walls and the portals overlap with the obstacles) but this overlap is harmless and will not impede the meshing and hence, the user can choose to ignore the overlaps for this case.

The system will also analyse the requirements for extended regions. Basically an extended region will be needed outside of any VENT object (the doors and windows in the low XY wall of the geometry). If there are multiple vents on a surface then only one extended region will be required for the whole of the surface. The system presents a selection window that shows the necessary and recommended extended region creation. The system may choose to recommend that some adjacent surfaces also have an extended region when the vents are in close proximity to the adjacent surfaces. This is necessary because the free surface patch applied to the outside of the extended region is an idealised boundary condition that actually applies a long way away from the vent. In practice we tolerate a separation in the range of 4 to 10 metres as this gives acceptable simulation behaviour. In the current scenario there is only one required extended region and the user should accept this choice without creating any additional extended regions. The Additional Extended Regions window appears as follows:

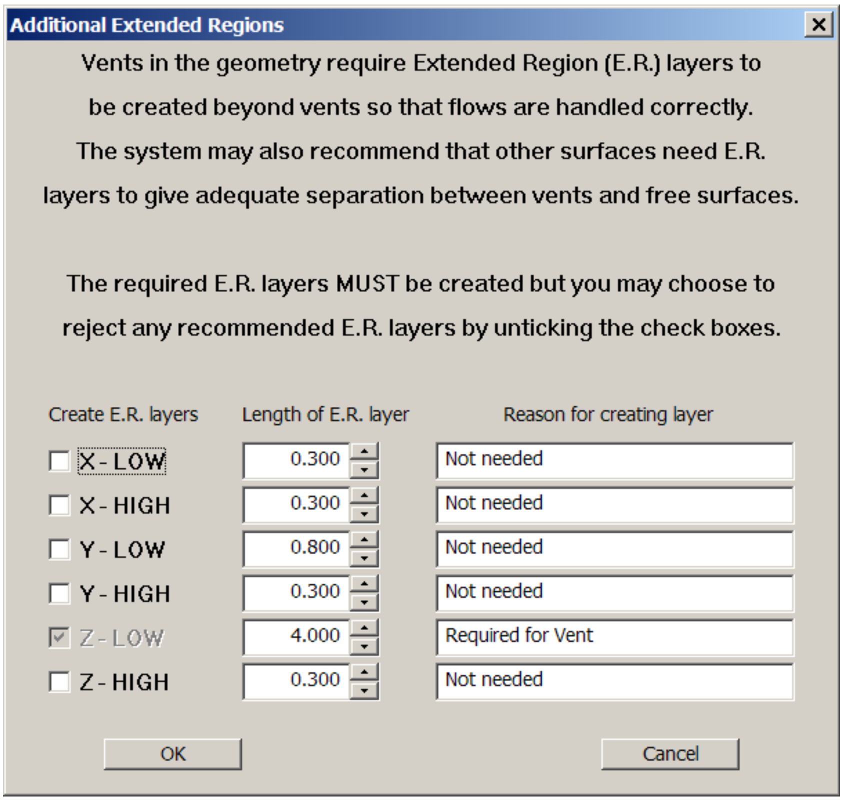

Figure 25-47 Window showing the options for creating extended regions.

Once all of the geometry and modelling issues have been resolved, the meshing system will analyse the geometry and present the Cell budget window. This has the automated meshing system's recommended cell budget with individual user-selectable cell budgets for each of the co-ordinate directions. It is recommended that the case is first meshed as a course mesh (usually a good option for checking that a case has been correctly specified and that all of the physics and numerics behave as expected). Unfortunately, for the course mesh, the meshing system struggles to create a very even mesh. The Cell Budget window appears as follows:

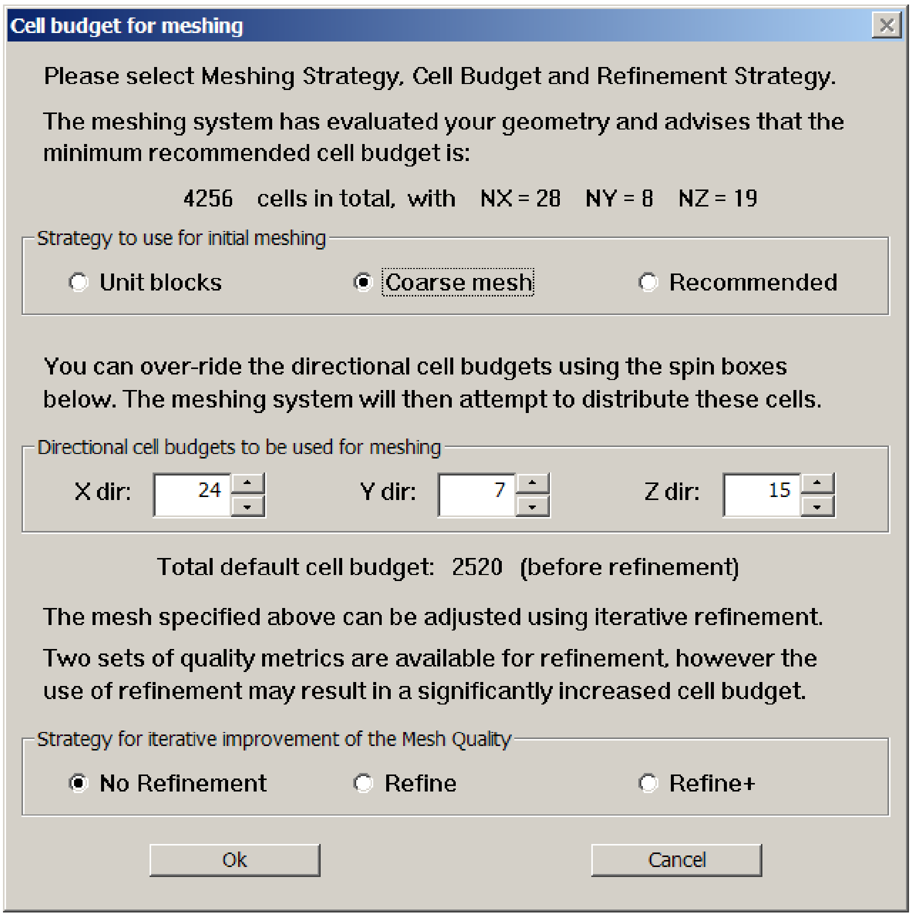

Figure 25-48 Cell budget for meshing.

The first attempt at meshing using a course mesh appears as follows:

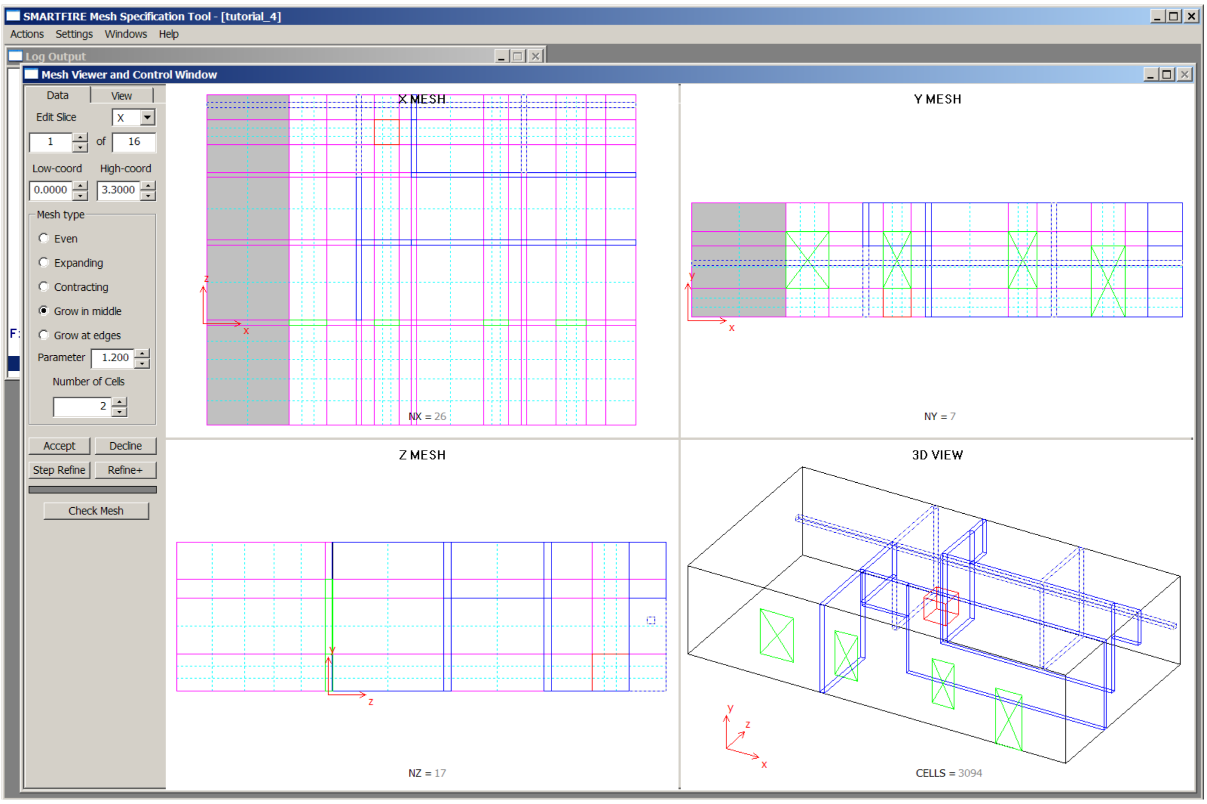

Figure 25-49 First attempt course mesh from the automated meshing system.

This mesh would NOT be acceptable for a production run, however it would allow a quick test run to be performed in order to check that the geometry is correct and that all events (for a transient simulation with transitional behaviour) work as required. In order to create a suitably even and more refined mesh, the number of cells in some of the mesh slices will need to be increased. This could be performed manually by selecting on the mesh windows to highlight a slice for refinement and then using the number of cells spin box to change the number of cells in that slice. Alternatively, the button labelled [Refine+] can be used to iteratively insert cells into the geometry slices until the mesh quality is deemed to be acceptable (or the cell budget is too large to run on the target hardware). Even a mesh that has been automatically refined might have areas that can be further improved so it is important to visually inspect the mesh to see if any further modifications are necessary.

It should be noted that there is no “absolutely correct mesh” for any simulation and, with experience, the user will eventually develop knowledge of when and where cells need to be refined. Generally the following points should be considered when refining a mesh.

(1) The fire will be generating a highly buoyant plume that may lean over as air is entrained or channelled around obstacles. It is good practice to provide additional cells in the vicinity of the fire to give the plume as much freedom as possible.

(2) Vents are likely to develop higher speed, channelled flows and will consequently benefit from having as many cells as possible across their height and width.

(3) Where a buoyant plume impacts a ceiling (or a ceiling jet impacts a wall) there should be as many cells as possible to allow for plume (or jet) rotation.

(4) It is not a good idea to have largely dissimilar sizes between adjacent mesh cells.

(5) It is not a good idea to have very squashed (or elongated) mesh cells.

After automatic refinement (using [Refine+]), the mesh should appear as follows:

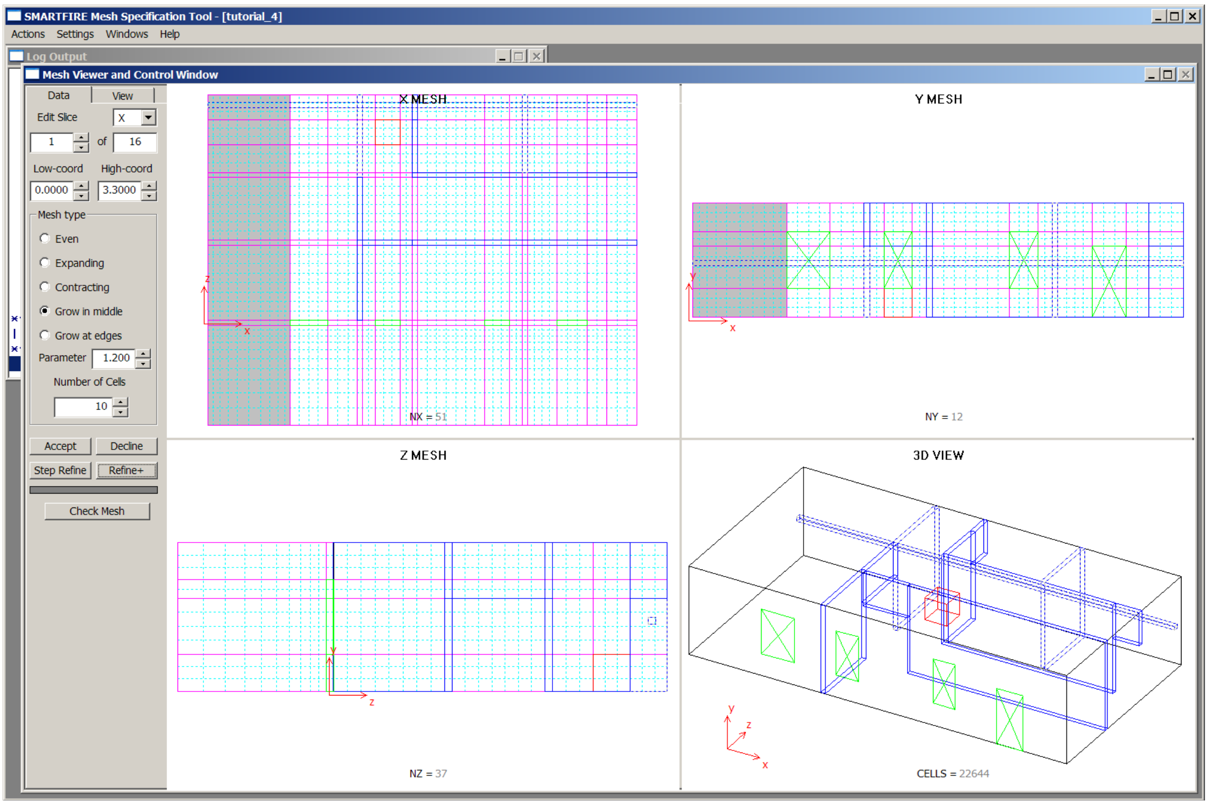

Figure 25-50 Refined mesh.

Note that the cell budget has increased dramatically from the first course meshing attempt, but that the refined mesh is much more evenly spaced in all three coordinate directions. The mesh is now ready and can be saved for simulation. Pressing the [Accept] button will allow the mesh and scenario specification to be saved as a set of specification files for the CFD Engine to read. Pressing the [Accept] button will also cause the automated meshing tool to close and control will then return to the ***SMARTFIRE*** Case Specification Environment.

25.5.4 STEP 3: RUNNING THE CFD ENGINE

The final stage involves performing the numerical simulation itself using the CFD engine component of ***SMARTFIRE***. To run the CFD engine select on the main menu item [Run] and the [Run CFD Engine] option. This will launch the numerical CFD engine that will automatically load the case specification and mesh that you have just created. You may have to be a little patient as this stage involves a considerable amount of file parsing, memory allocation and initialisation. Eventually the user interface for the CFD engine will appear completely. It should be noted that if you are using a fairly low resolution display (under 1024 x 768 pixels) then the CFD engine use-interface may display itself with more compact windows and a slightly different layout (e.g. with more of the windows initially closed) in order to create a usable display.

Once the CFD Engine has loaded the data and performed all of the require initialisations then the User Interface will be displayed with the un-processed view of the simulation case. The initial CFD Engine display will appear as follows:

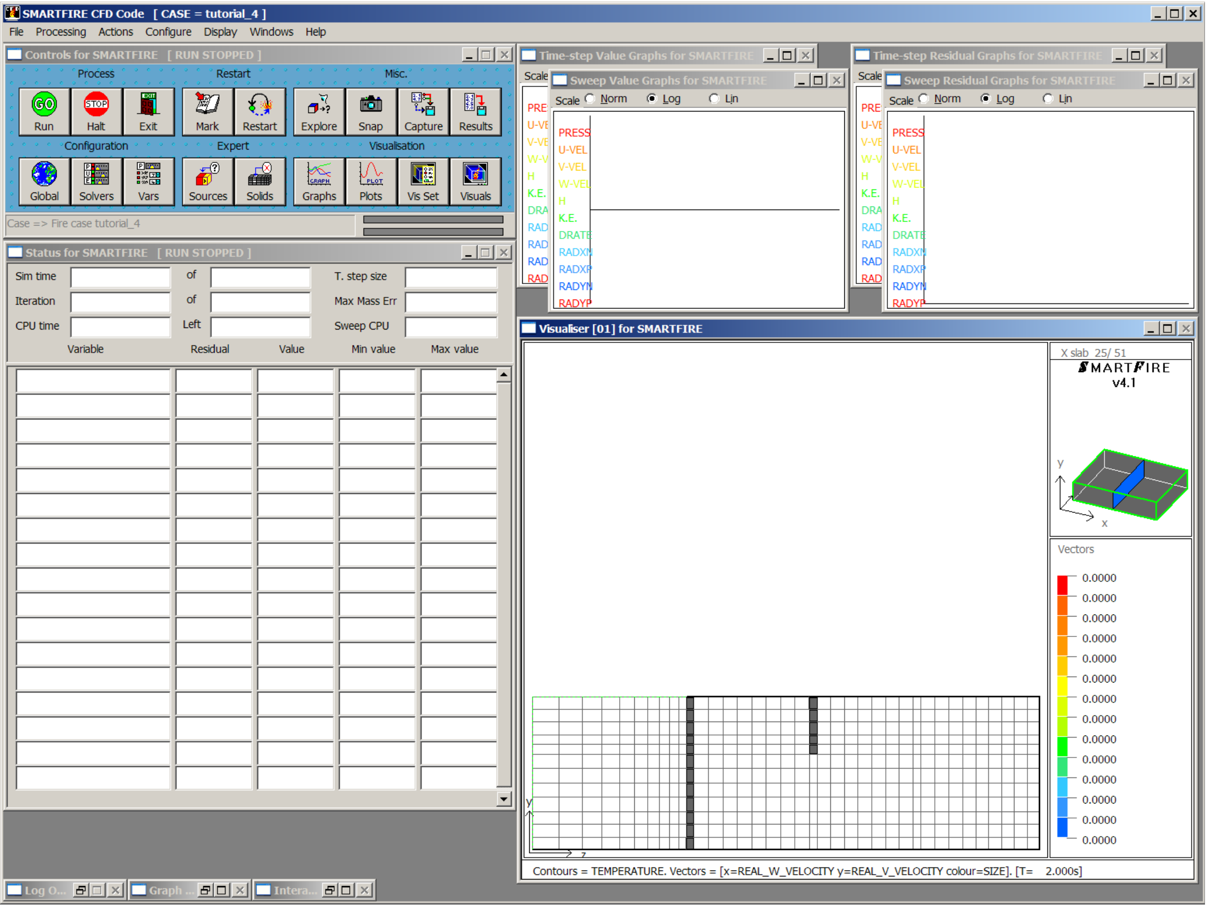

Figure 25-51 *SMARTFIRE* CFD Engine user interface at start up.

The Case Specification Environment will have pre-selected a default view plane for the visualisation (typically at the halfway layer of cells in the X-direction). Unfortunately this view plane might not be a particularly interesting one to display, so it is worth selecting a good plane to view throughout the simulation. In this case we have not kept any information about what would be a good plane to select but we can use the visual configuration window to "find" a suitable plane. If we press the [Visual] button then the Visual Configuration Window will open and we can use it to investigate the arrangement of the objects so that a more interesting slice plane can be found. By selecting the lowest Y slice, we can see where the fire is located in the Visual Display window.

It is then a simple matter of re-entering the Visual Configuration Window and selecting the planar slice that will give a more meaningful display of what is occurring in the vicinity of the fire.

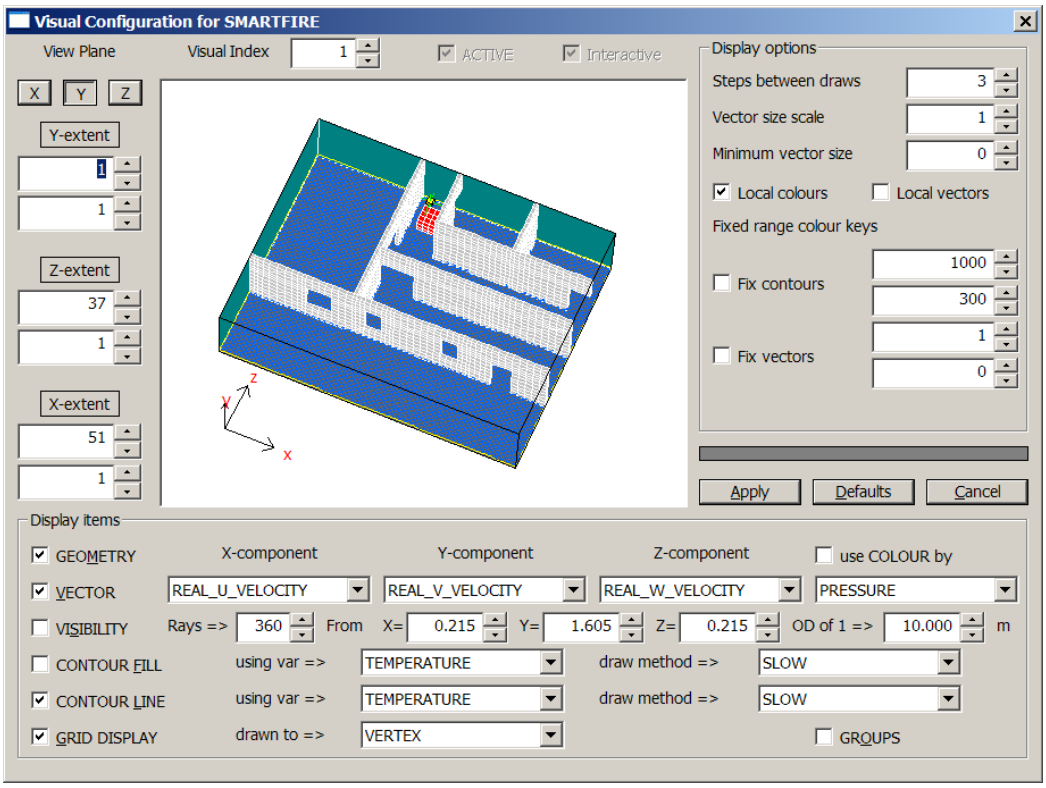

Figure 25-52 Visual Configuration selecting the lowest Y-layer of cells to view.

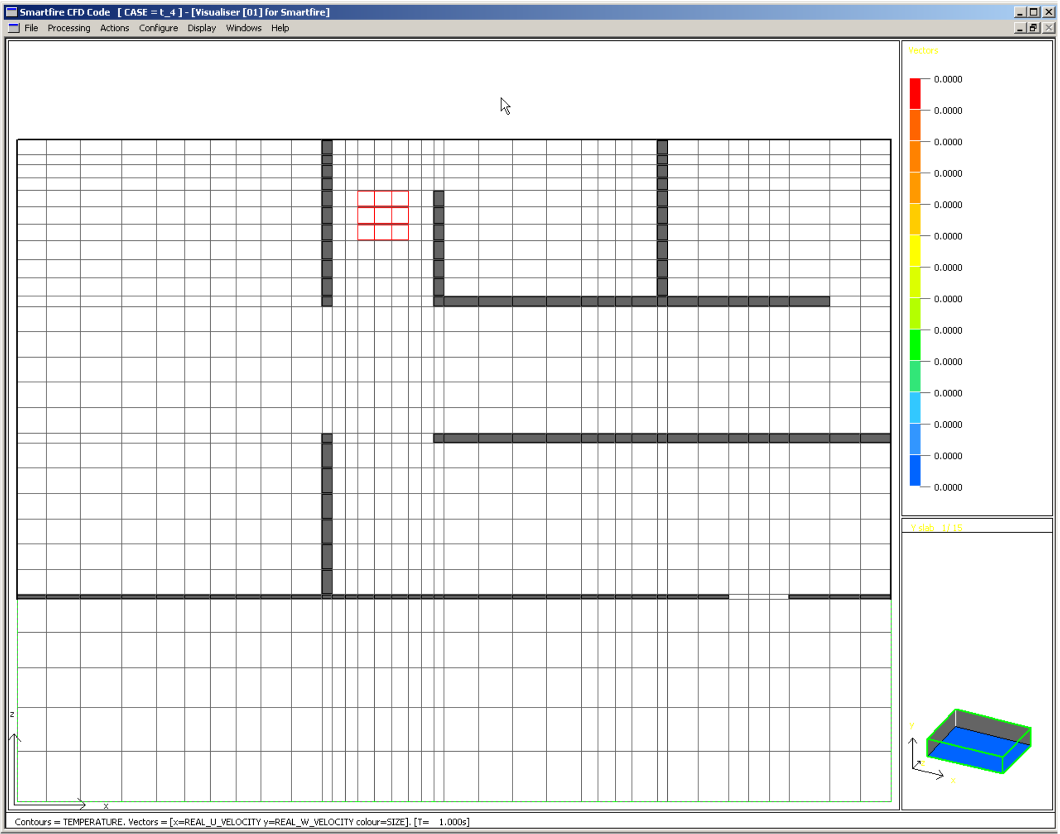

Figure 25-53 Visualisation of the lowest Y-slice of cells showing the fire location.

After selecting the X=22 slice (check this is through the centre of the fire as it will depend on the mesh that was used) the following Visual Display will appear on the User Interface.

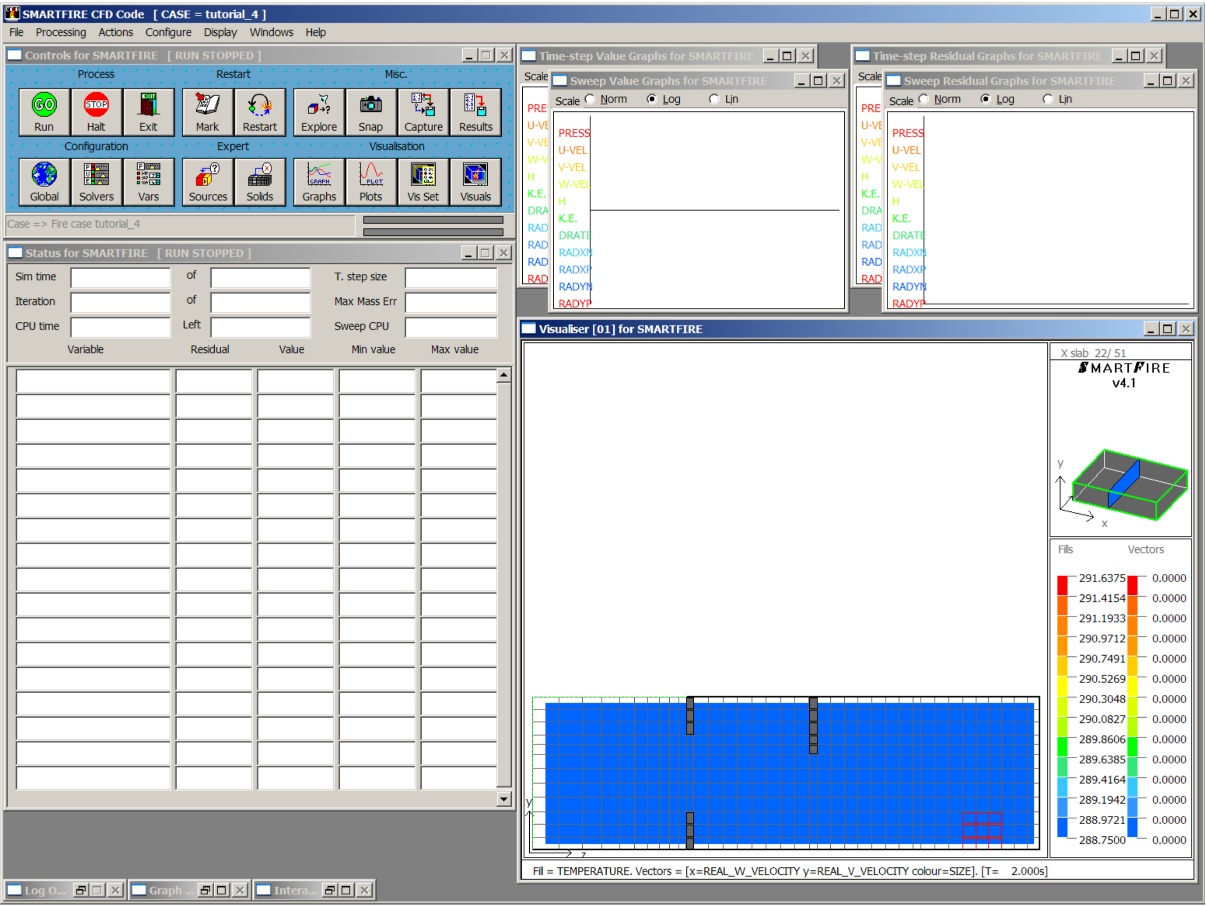

Figure 25-54 *SMARTFIRE* User Interface after selection of a more interesting visual slice plane through the fire.

It should be noted that the Visual Configuration Window could also be used to change the features that are displayed in the Visualizer. In the image above the [Contour Fill] option of "TEMPERATURE" has been activated because this gives a clearer representation of the temperatures in the fire plane, especially when using vectors of velocity in the same display.

Before starting the simulation, it is worth considering the forms and frequency of data capture from the simulation. If the user simply wishes to see what conditions are like at a particular time then the simulation can be run until that required simulation time ha been reached. In our scenario we are not sure what time the conditions will occur so we need to save sufficient data to allow us to analyse the time at which certain conditions were experienced. The user should open the [Configure] menu from the main menu and select [Data Capture]. This presents the Data Capture Configuration menu, where the frequency and nature of the data that will be saved during (and at the end of) the simulation, can be configured. In this simulation it is recommended that the user save the results, graphs and visuals at the end of every time step. This is configured by ticking the appropriate check boxes in the "Automatic Transient Outputs" section of the menu.

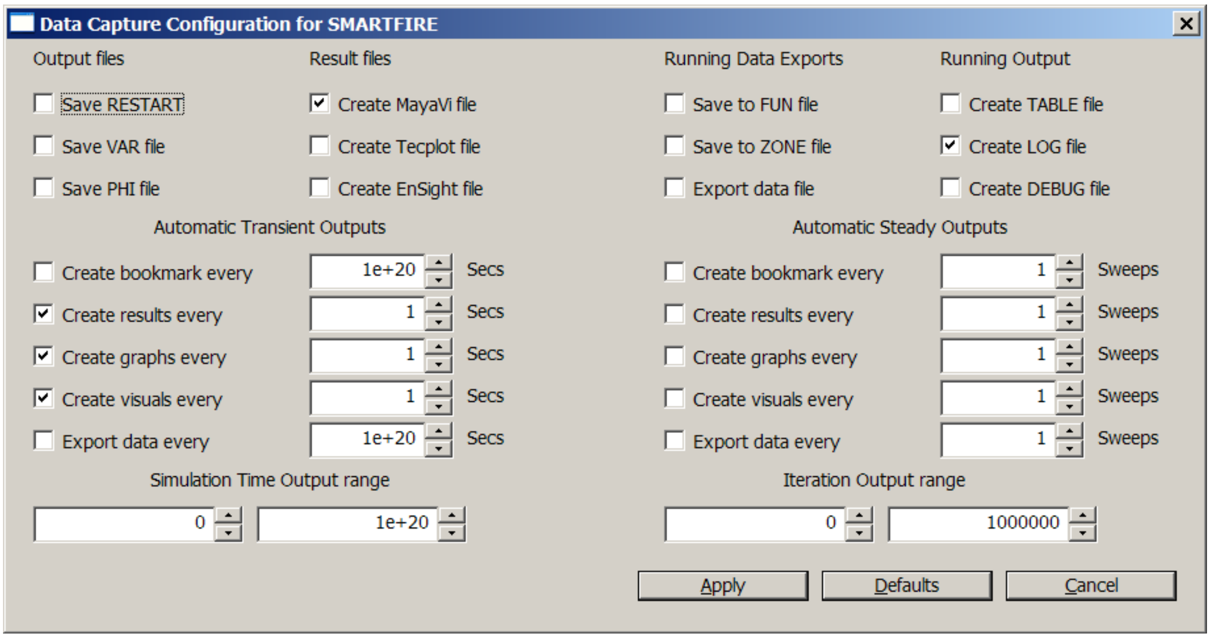

Figure 25-55 Data Capture Configuration menu.

Now that everything has been configured it is possible to start the processing of the simulation scenario.

In order to start the numerical simulation, you will need to press the [Run] button, marked with a GREEN (GO) icon. This will start the simulation process. To halt the simulation at any time, press the [Halt] button marked by a RED (STOP) icon. You can also terminate the simulation at any point by pressing the [Exit] button (showing a little door icon). There are a number of extra buttons and other controls that allow experts to intimately control the configuration of the solution process. However, non-experts are not recommended to make any changes to the control settings unless they are recommended in the user guide.

The graphical windows of the user interface present various views of the data and the status of the simulation during the simulation process. The things to look for are:

1) The residual graphs that are indicators of the convergence of the solved and calculated variables of the numerical simulation process (Top right).

2) The monitor values and variable residuals (current solution error states) of various important variables are shown in the status window (Bottom left).

3) The emerging vector flow and temperature contour patterns for the particular selected slice of the room in the Visualizer window (Bottom right). ***SMARTFIRE*** is now able to maintain multiple visualization windows, each containing completely independent view and display selections.

4) The data ranges for each variable displayed in the status window (Bottom left).

5) The control window has progress bars in its bottom right corner (below the visualisation buttons). These bars indicate solution progress. The upper bar is filled once every sweep whilst the lower bar is filled once for the whole configured

simulation (Top left).

6) The status window has displays indicating the sweep number and time step number (only for transient simulations) to indicate the current stage of processing (Bottom left).

7) The status window has estimates for the CPU time taken and remaining. These are only estimates but can give a reasonable approximation of the expected duration of a simulation (Bottom left).

8) A key feature of ***SMARTFIRE*** is the access to save a bookmark and restart from saved bookmarks at any time. The control button labelled [Mark] will drop a bookmark of the current stage of the solution into a database for this case. The button labelled [Restart] allows a previous bookmark state to be loaded as if subsequent processing had not happened. This can be invaluable for problematic simulations that need expert solution control or simply for saving data for future examination (Top left).

9) There is a control button labelled [Plots] that allows you to define line graphs through the data. These plot line graphs are updated as the solution progresses (Top left). There will be a single Plot window available to view that represents the monitor line that was created in the geometry specification.

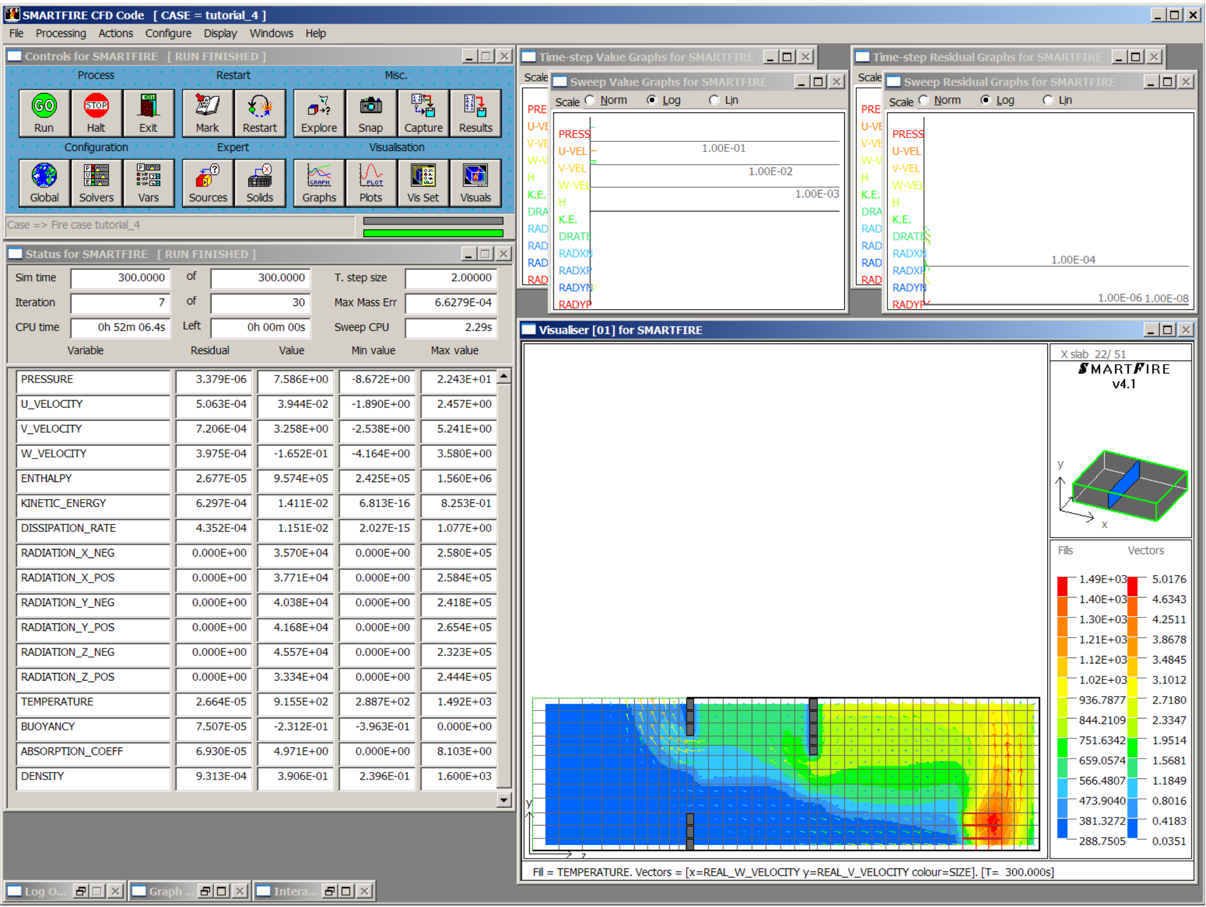

Figure 25-56 *SMARTFIRE* User Interface showing the end stage of the simulation.

25.5.5 STEP 4: INTERPRETING RESULTS FROM SMARTFIRE

If the simulation case has been run with a course mesh, it means that we cannot totally rely on the results for any safety critical assessment of the modelled scenario. Also we might not have not been very exacting about ensuring complete convergence (although a brief examination of the end of time step convergence history - the Time Step Residual Graph - indicates that most of the time steps ended at a reasonable state of convergence for this simulation). However we can use the results or data from the simulation to give a better understanding of the behaviour that we are likely to meet from this scenario.

The purpose of the simulation was to get a rough indication of the times before the rooms, neighbouring the fire, experienced conditions that are threatening to life. We are only considering the effect of elevated temperatures in the modelled scenario and effects such as smoke concentration, thermal radiation or the toxic gas concentration, are not considered. Again we can obtain a rough understanding of the fire effects by considering a typical harmful exposure time for a normal person to be approximately 1 minute at an air temperature of 180^{o}C. Remember that the SMARTFIRE CFD Engine is calculating temperatures in Kelvin. If we assume that, for safety considerations, reaching a critical temperature of 120^{o}C indicates the onset of dangerous - to human life - conditions. We can check through the plot graph data for the monitor line to determine at what time during the simulation each room experienced temperatures of 120^{o}C (i.e. 393K) or more in the vicinity of the monitor line.

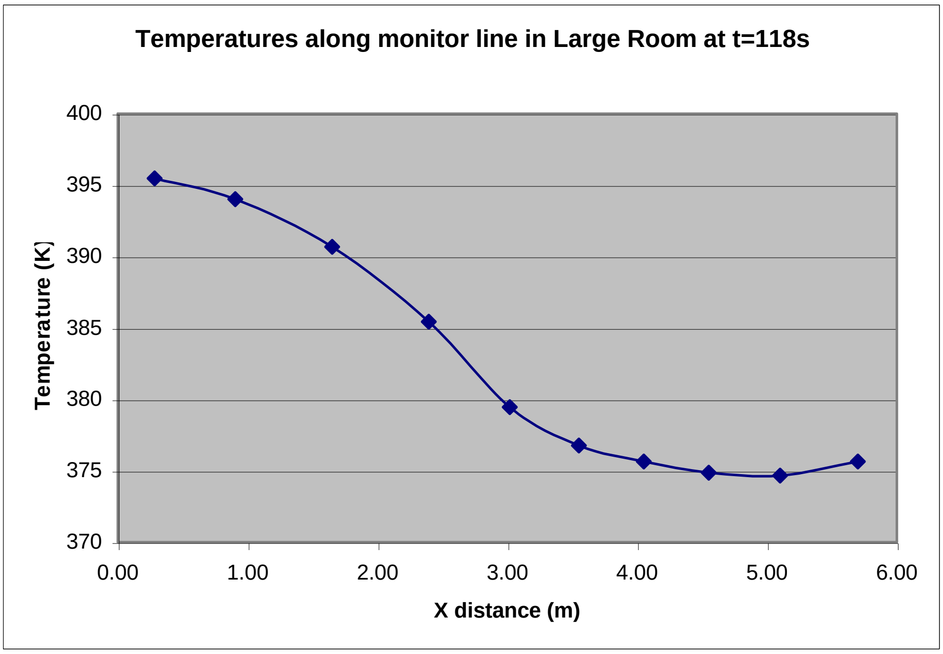

Figure 25-57 Graph of monitored temperatures in Large Room at t=118s.

Simple examination of the monitor line plot graph at the end of the simulation indicates that all of the monitored rooms are experiencing temperatures in excess of 500K (i.e. 227^{o}C).

Further analysis of the saved graph plot data files reveals the following times to reach the critical temperature.

Monitored Room	Time until first monitored cell above $T_{critical}$	Time until all monitored cells above $T_{critical}$
Large Room (with single window)	118s	126s
Small Room nearest to fire	78s	106s
Small Room furthest from fire	104s	116s

Table 25-58 Table of times to detect critical temperatures in monitored rooms.

At face value this suggests that (given our assessment of some critical temperature detected at a particular location as representative of the onset of dangerous conditions) the Large Room has approximately 2 minutes until the onset of dangerous conditions whilst the small room experiences dangerous conditions a little after 1 minute.

It should be noted that this tutorial is not intended to give an exacting study into the fire safety considerations pertaining to a single fire modelling scenario but, rather, give an indication of the nature of possible simulations and the types of questions that can be asked of the simulation results.

25.5.6 STEP 5: EXITING THE CFD ENGINE

To exit the code, you first need to exit the CFD engine interface using the [Exit] button. On normal termination, the ***SMARTFIRE*** CFD engine will save a number of files that can be used for further visual post-processing (abnormal termination will not save any files and is encountered when the main window [X] button is pressed. Finally the CFD engine user interface will close and you will get back to the original geometry set-up tool. If you want to save any changes you have made to the fire modelling case, select the [File] item and then the [Save] option from the main menu.

It is interesting to note that the case directory (for a completed simulation) contains many files that have been created during this exercise. Some of these files can be used for graphical post-processing and for re-starting this simulation from the stage when it was exited. Also there may be data capture files saved during your simulation.

This is the end of tutorial 4.

25.6 *TUTORIAL 5*

25.6.1 OVERVIEW

This tutorial extends the scenario used in Tutorial 4 to add a large, powerful, ducted extractor fan to the small room (nearest the fire). All of the other geometry and scenario specifications are identical to those in Tutorial 4 except for the changes to the geometry objects and the meshing that are needed for the inclusion of the fan. The plan view of the geometry is as follows:

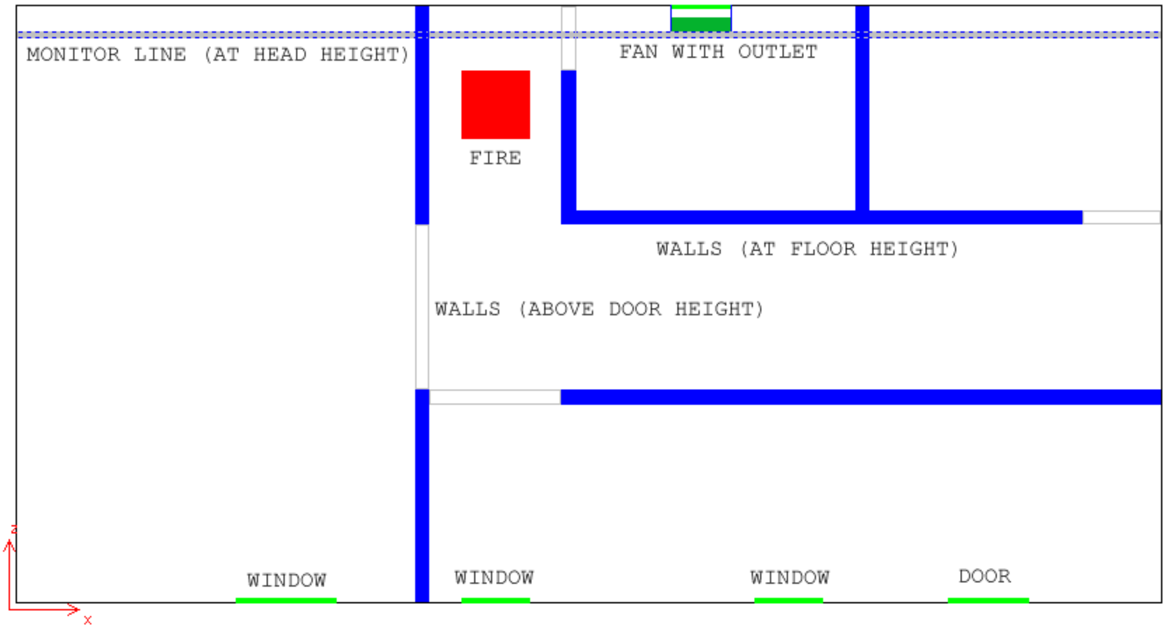

Figure 25-59 Plan view drawing showing the room layout geometry.

As with the Tutorial 4 scenario, certain assumptions have been made to simplify the specification as follows:

- The rooms are all assumed to be open during the fire simulation so all of the doorways and windows are fully open for all of the simulation.
- The doors on the end of the corridor are assumed closed throughout the simulation and they are of sufficient strength that the fire will not affect their structural integrity. There are actually windows present on the large outer wall of the largest room but these are assumed closed (and are hence not modelled) during the simulation (in practice this cuts down the requirement for an additional extended region and hence makes the simulation run faster). It should be noted that if the "closed" windows experience conditions that would make them break, then they would need to be modelled with the possibility of opening. Otherwise, imposing the always closed condition on the windows would create a false set of conditions for the simulation.

- Small air leaks around closed doors or windows are ignored and the heat transfer through the walls is considered to be negligible compared to heat transfer due to other processes, in the duration of the simulation.
- The corridor is assumed to have no combustible materials, which can support fire spread or secondary ignition.

The geometry of this case could be constructed from scratch but it is recommended that the user actually loads the ***SMARTFIRE*** model file created for Tutorial 4, saves it as a new case and then adds the extra objects required for the fan.

Again the fire is assumed to be a cleaning trolley fire at one end of the corridor. The fire reaches a peak heat release rate of 2MW in two minutes. Since the fire is in the corridor and will prevent conventional means of egress from the building, we are actually interested in determining the times to reach a critical temperature (at about head height) in the neighbouring rooms where there are no exterior doorways to use for exit.

The Fan (actually a SIMPLE FAN object) is assumed to be a smoke extraction system that is activated 20 seconds after the fire has been detected. For the sake of simplicity, it is assumed that the detection system is near to the fire source (in the corridor) and that the Fan is activated 20 seconds into the simulation. The Fan system is assumed to be sufficiently robust so that it will allow the extraction of very hot gasses for the duration of the simulation. This is not an unreasonable assumption if the fan motor and blades were well removed from the fire source by ducting and the fan has been designed with this task in mind. It is also assumed that all the ductwork for the fan is also sufficiently robust to maintain its integrity for the duration of the simulation. It should be noted that, in practice, the details of the failure characteristics of the ductwork and the fan unit would need to be taken into consideration and compared with the likely conditions experienced in the simulation.

25.6.2 STEP 1: LOADING THE BASE CASE AND RENAMING

Run the ***SMARTFIRE*** case specification tool. Once the graphical interface has opened, use [File] and [Open] to open the file loading menu. Use the file loading dialog window to browse to the folder containing the tutorial case from Tutorial 4 (probably called “tutorial_4” or "t_4"), typically found in the "smartfire\work" folder. Select the ***SMARTFIRE*** model file for this case (named "t_4.smf" - if using the naming convention suggested in Tutorial 4), and select the [Open] button. The base case from Tutorial 4 will be loaded and the default 3D wire frame view of the case will be displayed.

In order to prevent any subsequent geometry or scenario modifications from being accidentally written back into the Tutorial 4 case, it is recommended that the case is immediately renamed by selecting the [File] and [Save As] option. Use the file save dialog to browse back up to the "smartfire\work" folder - since you will currently be INSIDE the "t_4" folder, and enter the name "t_5" in the File name field and select the [Save] button. The case will be renamed and a newly named "t_5.smf" model file will be saved in a new "t_5" folder.

Now you can proceed to add the objects needed to create the simple fan.

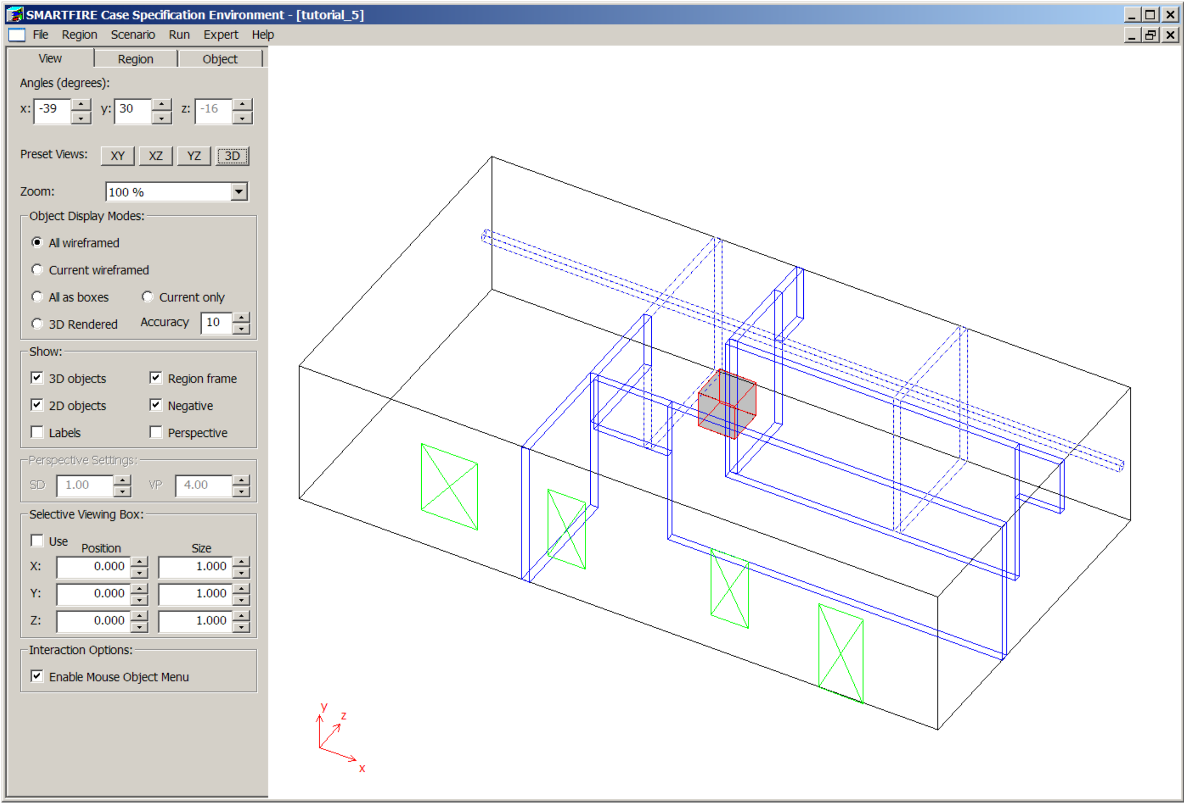

Figure 25-60 Specification tool showing the renamed case from Tutorial 4.

25.6.3 STEP 2: ADDING A DUCTED FAN TO THE SCENARIO

The actual SIMPLE FAN object is a generic fan source object that can be used to create forced ventilation ranging from ducted extractor fans to simple (unidirectional) desk fans. Constructing a ducted extract fan is a little more complex than simply adding the generic SIMPLE FAN object since there are additional considerations to be made in order to get the desired flow behaviour.

The SIMPLE FAN object needs some surrounding ducting to make it a "ducted" extractor fan. Otherwise flow would be able to enter (or leave) the fan unit from the sides and the fan would behave more like a simple desktop fan. This ducting could be constructed from four THIN PLATE objects or from four OBSTACLE objects depending on the best geometric representation of the actual ducting.

The fan unit is extracting through a solid wall but we are not actually concerned with the air once it has left the region of simulation. In order to achieve this effect it is necessary to place an OUTLET patch on the wall surface within the fan duct. This outlet patch provides a surface where all of the air extracted from the region can be dumped.

The final consideration is that the SIMPLE FAN object needs to spread the momentum forces over a layer of cells on either side of the SIMPLE FAN object itself. For this reason there must be at least a one cell air gap between the SIMPLE FAN object and the OUTLET patch. The following diagram should help to clarify the construction of the ducted extractor fan.

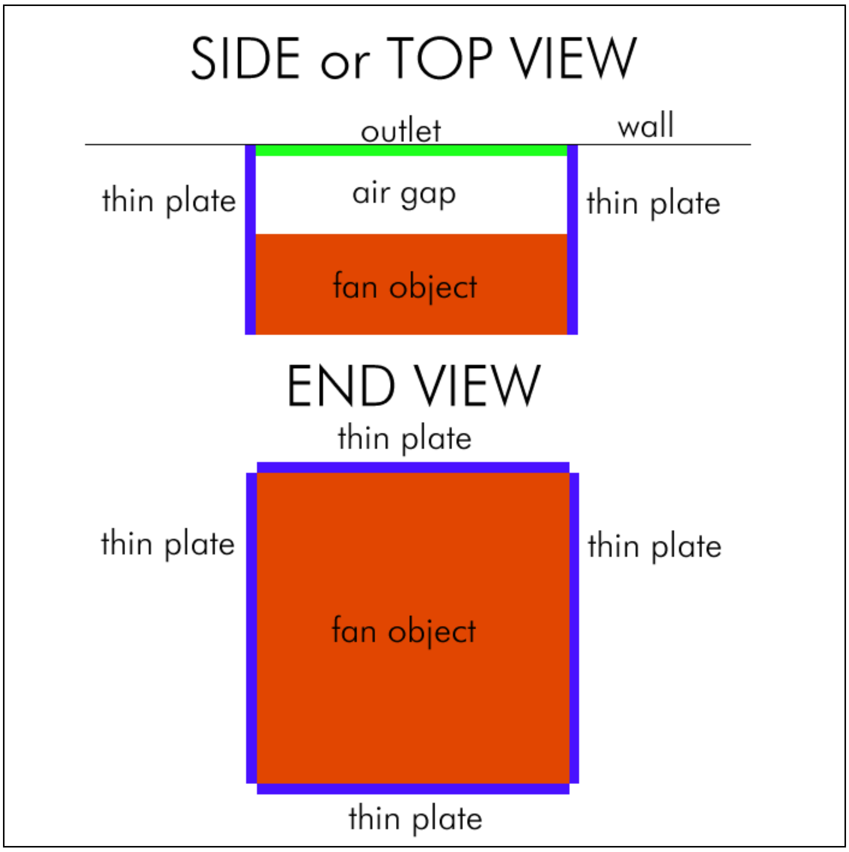

Figure 25-61 Components used for the construction of a ducted fan using a SIMPLE FAN object, an OUTLET object and four THIN PLATE objects.

The following objects define the ducted fan unit and should be added with the specified positions and sizes.

Outlet through the wall (in the small room nearest the fire):
OUTLET at (x=10.0, y=2.5) on the high-z wall. Size is (dx=0.8, dy=0.8). The OUTLET Pressure should be set to 0.0 Pa and the Temperature set to ambient (i.e. 288.15 K).

Fan blower (in the small room nearest the fire):
SIMPLE FAN at (x=10.0, y=2.5, z=8.6) on the low-z wall. Size is (dx=0.8, dy=0.8, dz=0.2). The Simple Fan blowing direction should be set to Z+, which is the outward direction for the high-z surface of the region - where the fan is located. The Fan velocity should be set to 5 m/s and the coefficient value left at the default 1000.0. The Start time for the fan should be set in the activation section so that the Fan will be “Active from start (added at start time)” with 20.0 seconds as the fan start time.

The SIMPLE FAN properties window should appear as follows:

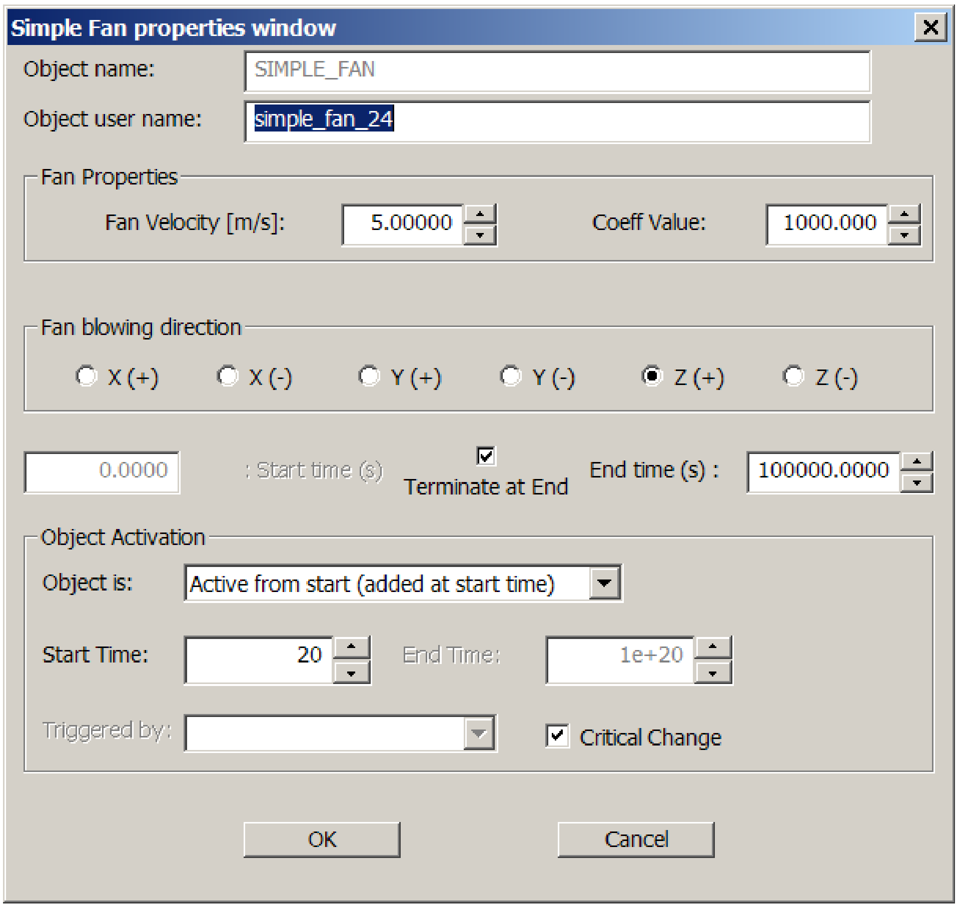

Figure 25-62 Simple Fan properties window.

Fan duct, low-x edge (in the small room nearest the fire):
THIN PLATE at (x=10.0, y=2.5, z=8.6) in the x-direction. Size is (dy=0.8, dz=0.4).

Fan duct, high-x edge (in the small room nearest the fire):
THIN PLATE at (x=10.8, y=2.5, z=8.6) in the x-direction. Size is (dy=0.8, dz=0.4).

Fan duct, low-y edge (in the small room nearest the fire):
THIN PLATE at (x=10.0, y=2.5, z=8.6) in the y-direction. Size is (dx=0.8, dz=0.4).

Fan duct, high-y edge (in the small room nearest the fire):
THIN PLATE at (x=10.0, y=3.3, z=8.6) in the y-direction. Size is (dx=0.8, dz=0.4).

The following images may help with the positioning and orientation of the objects used to create the fan unit.

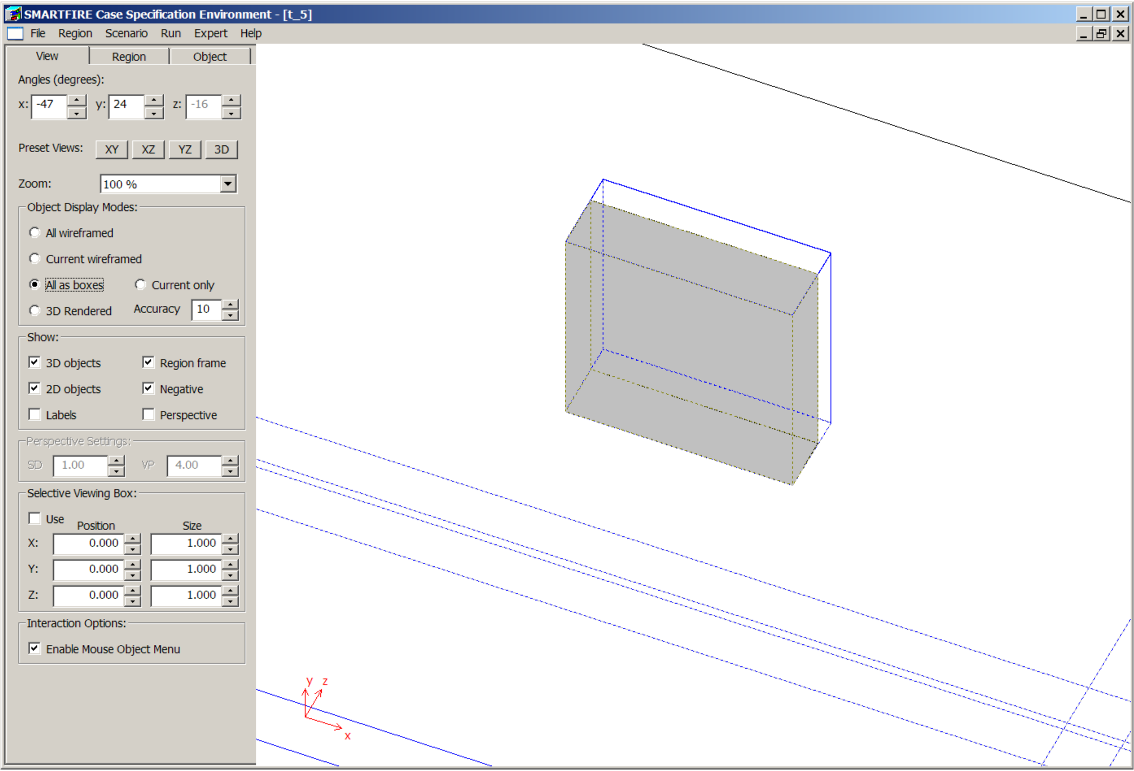

Figure 25-63 Close up wire frame view of the Fan unit.

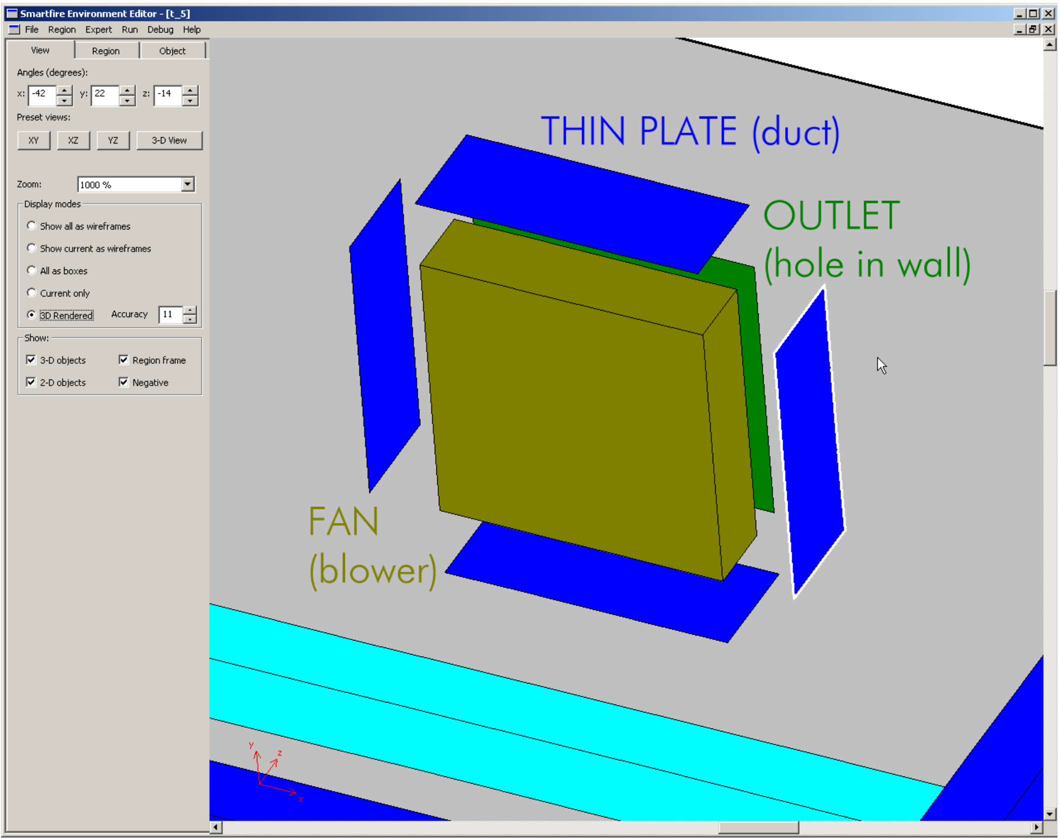

Figure 25-64 Exploded view (with components separated) of the Fan unit. Note that there is an air gap needed between the SIMPLE FAN object and the OUTLET object.

All of the other settings for the scenario are the same as for Tutorial 4, except that the [Enhanced Body Force] needs to be activated from the Problem Type window. It should be noted that the ***SMARTFIRE*** Case Specification Environment will detect that SIMPLE FAN and/or FAN objects have been used and will inform the user that the Enhanced Body Force must be activated when using fans.

25.6.4 STEP 3: CREATING A MESH FOR THE SIMULATION

Once you are satisfied with the geometry and physics handling of the case you need to run the mesh creation system. To run the automated meshing tool from the main menu, choose the [Run] option and the select [Create Mesh]. This will open the mesh creation tool user interface. The automated meshing tool will first determine if the current geometry already has an existing mesh previously created. If one is available then the user will be given the option to load the existing mesh or to create a new one. It is assumed that no existing mesh is available or that the user will create a new mesh.

It should be noted that the mesh created for this scenario, whilst similar to that created for Tutorial 4, will have differences in the number (and usage) of slices and the requirements for meshing will be different in the vicinity of the Fan. The addition of even a few extra objects can radically affect the overall mesh cell budget, since each object brings an extra 2 edges in each direction.

Once meshing is initiated, the meshing tool will check for any geometry or modelling issues that might need to be resolved. If there are any problems then the system will ask the user to select an appropriate resolving action. For example the system WILL detect overlapping objects in the geometry that has been specified above (the walls and the monitor line object overlap since the monitor line extends from end to end of the region passing through some of the walls) but this overlap is harmless and will not impede the meshing and hence, the user can choose to ignore the overlaps for this case.

The system will also analyse the requirements for extended regions. Basically an extended region will be needed outside of any VENT object (the doors and windows in the low XY wall of the geometry). If there are multiple vents on a surface then only one extended region will be required for the whole of the surface. The system presents a selection window that shows the necessary and recommended extended region creation. The system may choose to recommend that some adjacent surfaces also have an extended region when the vents are in close proximity to the adjacent surfaces. This is necessary because the free surface patch applied to the outside of the extended region is an idealised boundary condition that actually applies a long way away from the vent. In practice we tolerate a separation in the range of 4 to 10 metres as this gives acceptable simulation behaviour. In the current scenario there is only one required extended region and the user should accept this choice without creating any additional extended regions. The Additional Extended Regions window appears as follows:

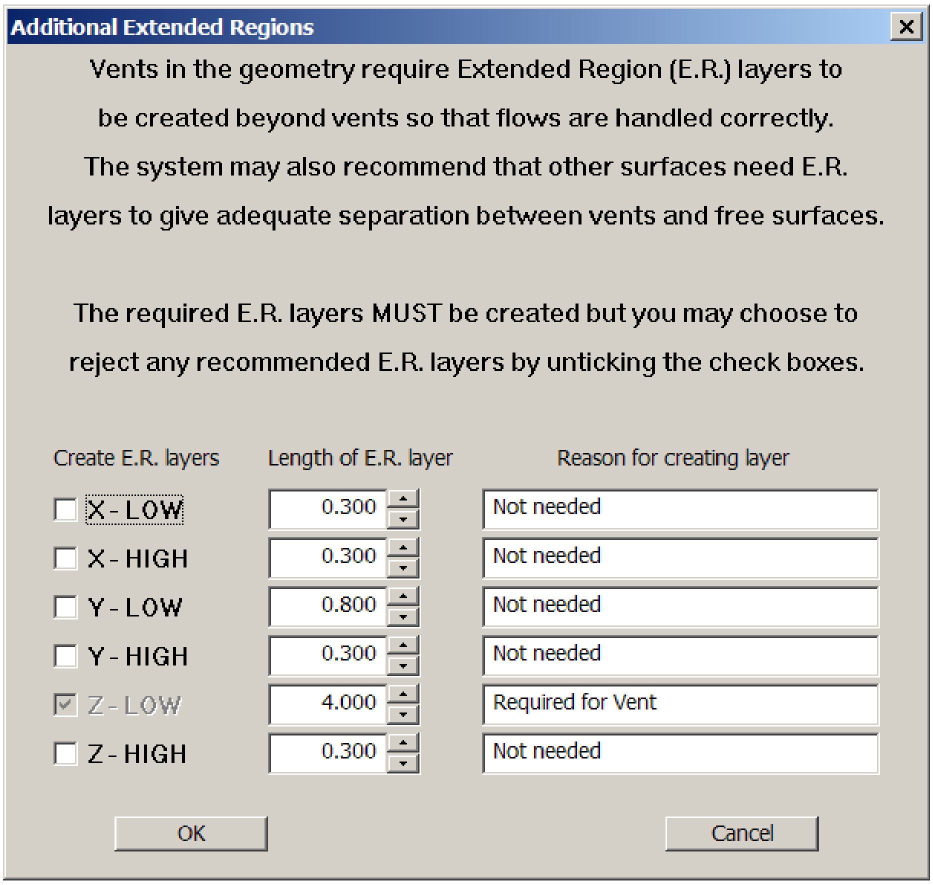

Figure 25-65 Window showing the options for creating extended regions.

Once all of the geometry and modelling issues have been resolved, the meshing system will analyse the geometry and present a Cell budget window. This has the automated meshing system's recommended cell budget with individual user-selectable cell budgets for each of the co-ordinate directions. It is recommended that this case be meshed as a unit block mesh with the refine option. This creates a reasonable mesh to run this case although it might not be appropriate for a production run, where accurate results are needed. The Cell Budget window appears as follows:

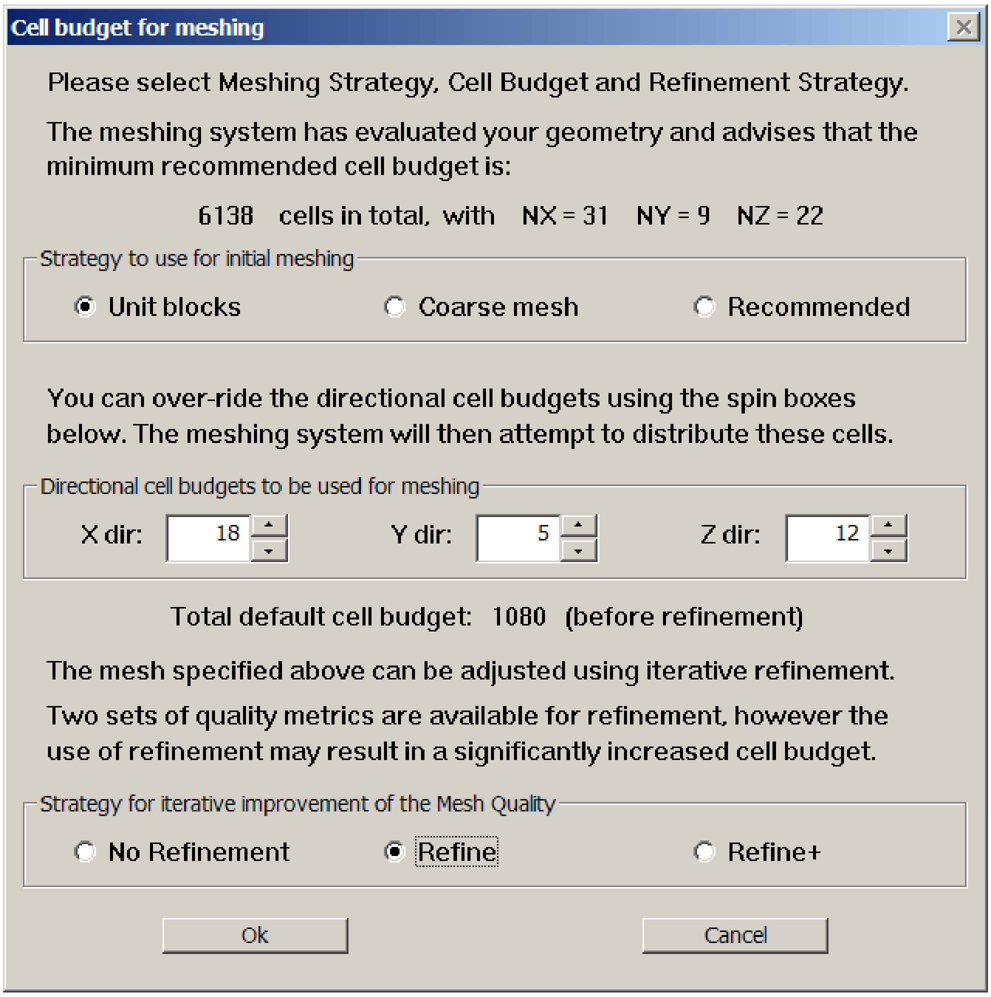

Figure 25-66 Cell budget for meshing.

This attempt at meshing using a refined mesh, appears as follows:

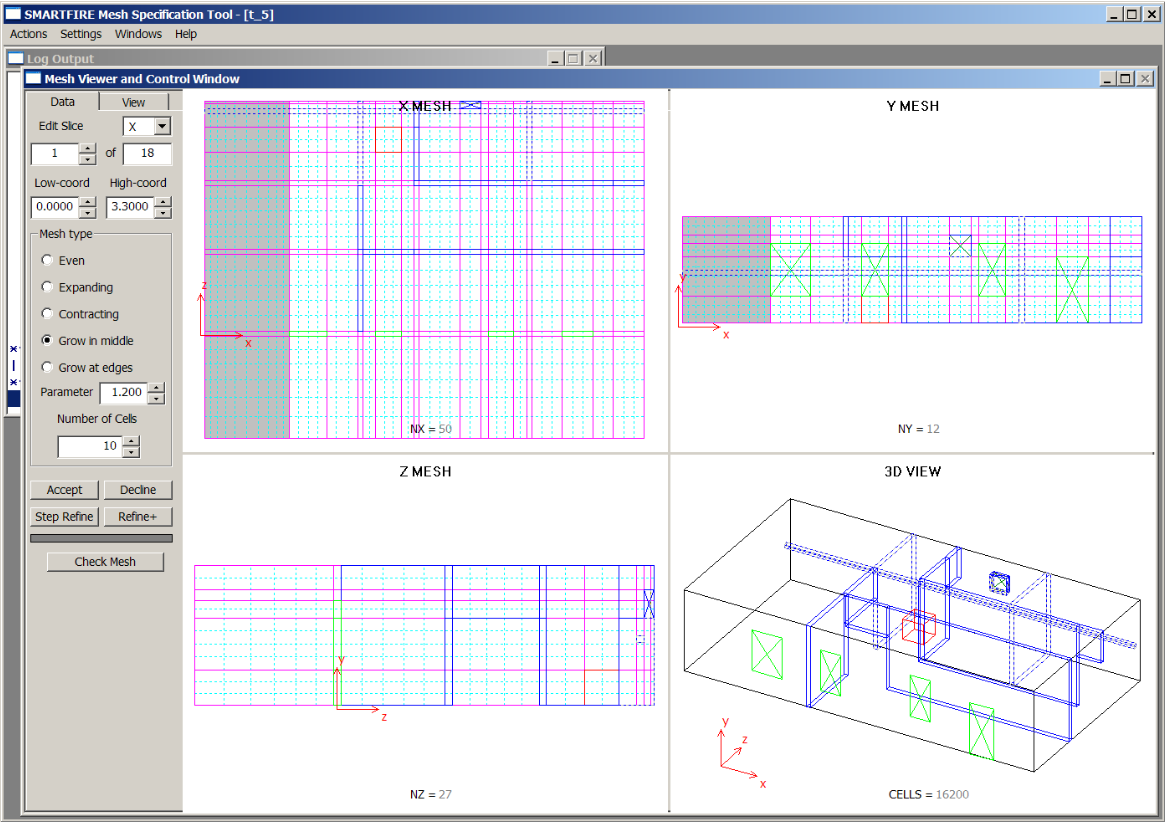

Figure 25-67 Mesh from the automated meshing system.

This mesh would be adequate for a tutorial run to check that the scenario is working as expected, however a production run – where the results need to be accurate – would need a more even and higher quality mesh. It should be noted that it is possible to manually refine the mesh using the "slice" selection combo box and slice number spin box to highlight a slice for refinement or by pointing at the required slice in the display windows and selecting (with the mouse pointer and button) to make the current slice active (highlighted with grey=internal region or green=external region shading).

The user should always visually check the mesh after it has been refined to ensure that there are no issues caused by irregular slices or having a restricted cell budget. It should be noted that there is no absolutely correct mesh for any simulation and, with experience, the user will eventually develop knowledge of when and where cells need to be refined. Generally the following points should be considered when refining a mesh.

(1) The fire will be generating a highly buoyant plume that may lean over as air is entrained or channelled around obstacles. It is good practice to provide additional cells in the vicinity of the fire to give the plume as much freedom as possible.

(2) Vents are likely to have higher speed channelled flows and will consequently benefit from having as many cells as possible across their height and width.

(3) Where a buoyant plume impacts a ceiling (or wall) there should be as many cells as possible to allow for plume rotation into ceiling jets.

(4) It is not a good idea to have largely dissimilar sizes between adjacent mesh cells.

(5) It is not a good idea to have very squashed (or elongated) mesh cells.

(6) The flows in the vicinity of a Fan are likely to be complex and quite strong. It is advisable to have reasonable refinement of cells between openings (e.g. the doorway) and the Fan.

The mesh is ready and can be saved for simulation. Pressing the [Accept] button will allow the mesh and scenario specification to be saved as a set of specification files for the CFD Engine to read. Pressing the [Accept] button will also cause the automated meshing tool to close and control will then return to the ***SMARTFIRE*** Case Specification Environment.

25.6.5 STEP 4: RUNING THE CFD ENGINE

The final stage involves performing the numerical simulation itself using the CFD engine component of ***SMARTFIRE***. To run the CFD engine select on the main menu item [Run] and the [Run CFD Engine] option. This will launch the numerical CFD engine that will automatically load the case specification and mesh that you have just created. You may have to be a little patient as this stage involves a considerable amount of file parsing, memory allocation and initialisation. Eventually the user interface for the CFD engine will appear completely. It should be noted that if you are using a fairly low resolution display then the CFD engine use-interface may display itself with more compact windows and a slightly different layout (e.g. with more of the windows initially closed) in order to create a usable display.

Once the CFD Engine has loaded the data and performed all of the required initialisations, then the User Interface will be displayed with the un-processed view of the simulation case. The initial CFD Engine display will appear as follows:

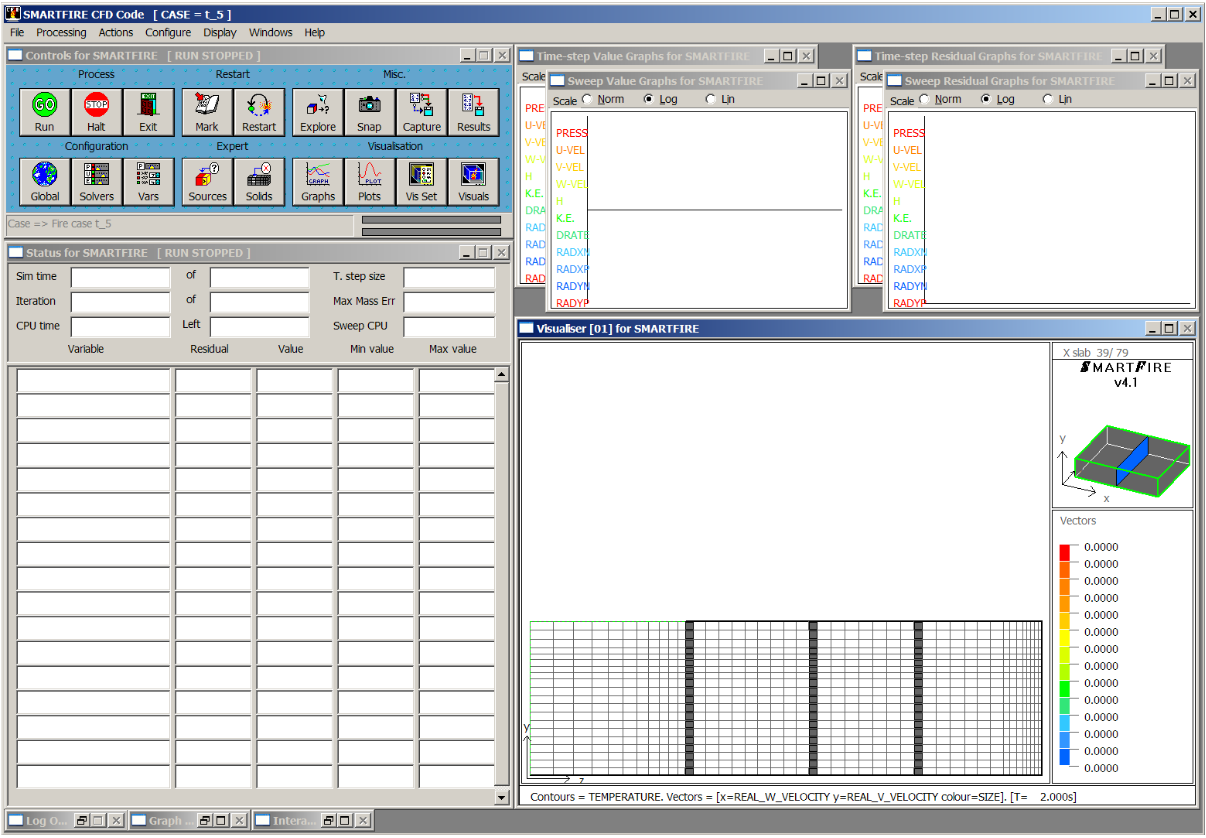

Figure 25-68 *SMARTFIRE* CFD Engine user interface at start up.

The Case Specification Environment will have selected a default view plane for the visualisation (typically at the halfway layer of cells in the X-direction). Unfortunately this view plane might not be a particularly interesting one to display, so it is worth selecting a good plane to view throughout the simulation. In this case we have not kept any information about what would be a good plane to select but we can use the visual configuration window to "find" a suitable plane. If we press the [Visual] button then the Visual Configuration Window will open and we can use it to investigate the arrangement of the objects so that a more interesting slice plane can be found. By selecting the lowest Y slice, we can see where the fire is located in the Visual Display window.

It is then a simple matter of re-entering the Visual Configuration Window and selecting the planar slice that will give a more meaningful display of what is occurring in the vicinity of the fire.

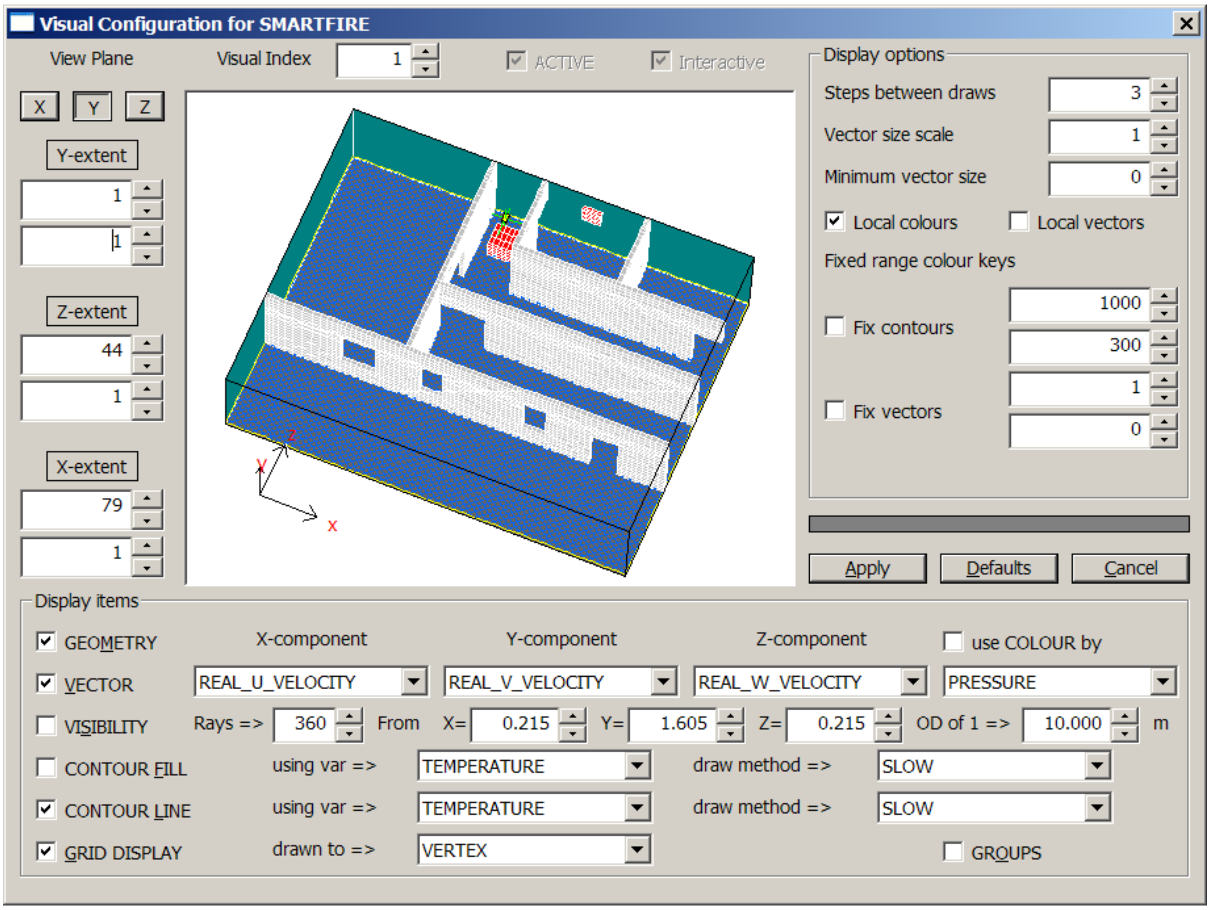

Figure 25-69 Visual Configuration selecting the lowest Y-layer of cells to view.

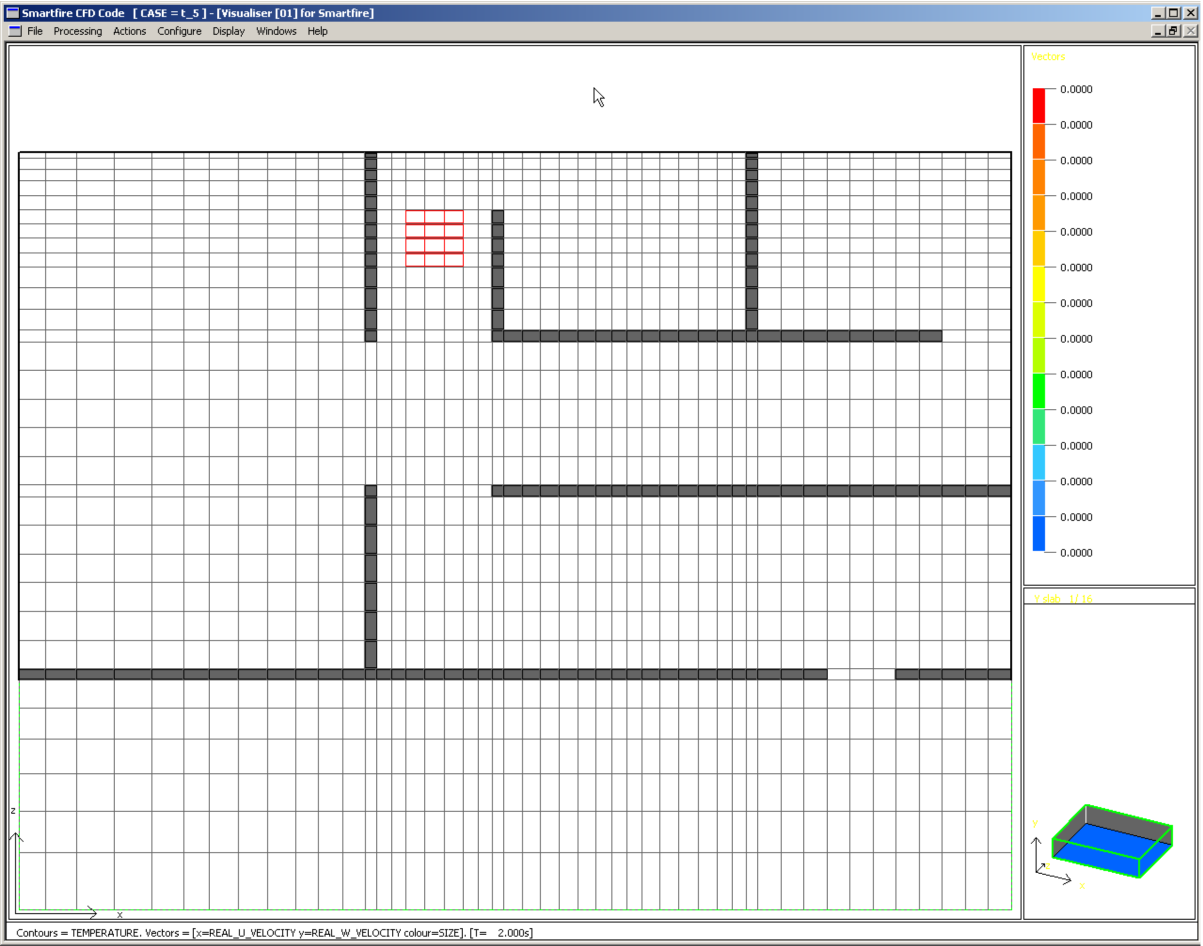

Figure 25-70 Visualisation of the lowest Y-slice of cells showing the fire location.

In order to keep track of both the Fire plane and the Fan plane, two visualizations are configured using the [Visual Index] spin box in the Visual Configuration Window and the [Active] check box. Visual [1] depicts the X=32 slice (the user should check that this is through the centre of the fire) and Visual [2] depicts the X=44 slice (for this mesh the cut plane is through the centre of the Fan). Using these two visualizations and manually arranging the Visualizer windows so that both are visible on screen, makes the User Interface appear as follows:

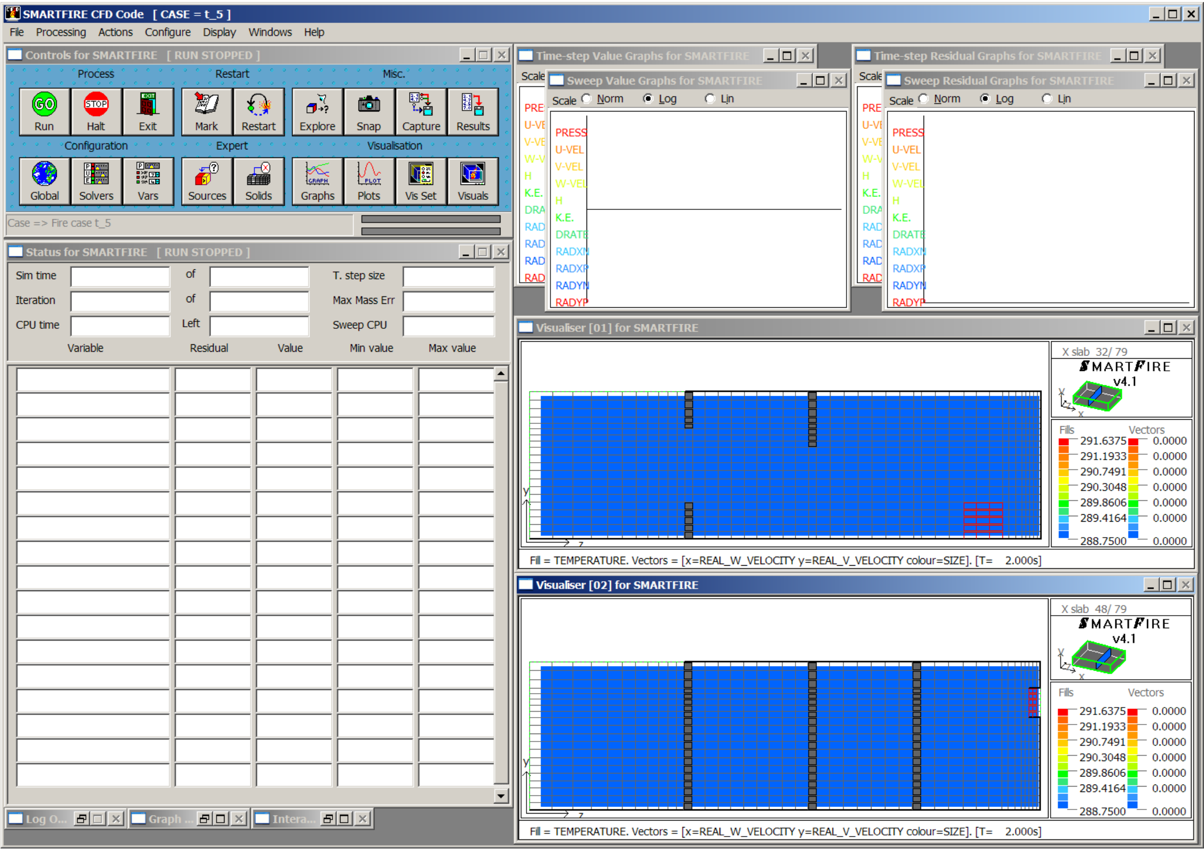

Figure 25-71 *SMARTFIRE* showing visuals of the fire plane and the fan plane.

It should be noted that the Visual Configuration Window could also be used to change the features that are displayed in the Visualizers. In the image above the [Contour Fill] option of "TEMPERATURE" has been activated because this gives a clearer representation of the temperatures in the fire plane, especially when using vectors of velocity in the same display.

Before starting the simulation it is worth considering the forms and frequency of data capture from the simulation. If the user merely wishes to see what conditions are like at a particular time then it is simple to run the simulation until that required simulation time is reached. In our scenario we are not sure what time the conditions will occur so we need to save sufficient data to allow us to analyse the time at which certain conditions were experienced. The user should open the [Configure] menu from the main menu and select [Data Capture]. This presents the Data Capture Configuration menu, where the frequency and nature of the data that will be saved during (and at the end of) the simulation, can be configured. In this simulation it is recommended that the user save the results, graphs and visuals at the end of every time step. This is configured by ticking the appropriate check boxes, and setting the times between saves, in the "Automatic Transient Outputs" section of the menu.

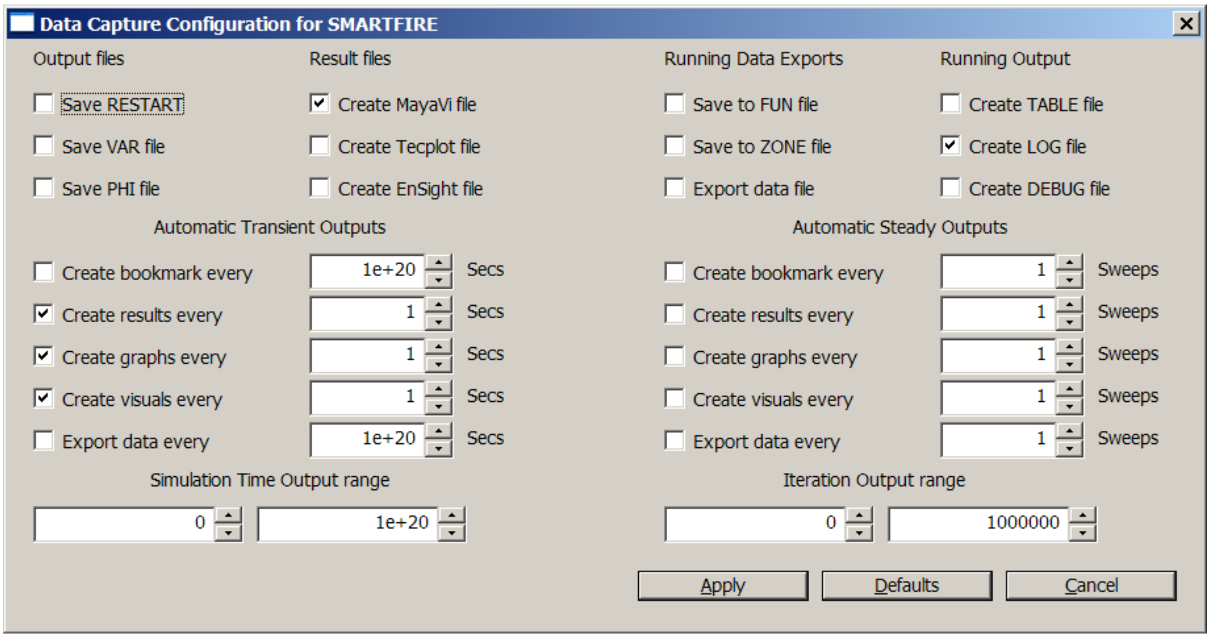

Figure 25-72 Data Capture Configuration menu.

Now that everything has been configured, it is possible to start the processing of the simulation scenario.

In order to start the numerical simulation, you will need to press the [Run] button, marked with a GREEN (GO) icon. This will start the simulation process. To halt the simulation at any time, press the [Halt] button marked by a RED (STOP) icon. You can also terminate the simulation at any point by pressing the [Exit] button (showing a little door icon). There are a number of extra buttons and other controls that allow experts to intimately control the configuration of the solution process. However, non-experts are not recommended to make any changes to the control settings unless they are recommended in the user guide.

The graphical windows of the user interface present various views of the data and the status of the simulation during the simulation process. The things to look for are:

1) The residual graphs that are indicators of the convergence of the solved and calculated variables of the numerical simulation process (Top right).

2) The monitor values and variable residuals (current solution error states) of various important variables are shown in the status window (Bottom left).

3) The emerging vector flow and temperature contour patterns for the particular selected slice of the room in the Visualizer window (Bottom right). ***SMARTFIRE*** is now able to maintain multiple visualization windows, each containing completely independent view and display selections.

4) The data ranges for each variable displayed in the status window (Bottom left).

5) The control window has progress bars in its bottom right corner (below the visualisation buttons). These bars indicate solution progress. The upper bar is filled once every sweep whilst the lower bar is filled once for the whole simulation.

6) The status window has displays indicating the sweep number and time step number (only for transient simulations) indicating the current stage of processing.

7) The status window has estimates for the CPU time taken and remaining. These are only estimates but can give a reasonable approximation of the expected duration of a simulation (Bottom left).

8) A key feature of ***SMARTFIRE*** is the access to save a bookmark and restart from saved bookmarks at any time. The control button labelled [Mark] will drop a bookmark of the current stage of the solution into a database for this case. The button labelled [Restart] allows a previous bookmark state to be loaded as if subsequent processing had not happened. This can be invaluable for problematic simulations that need expert solution control or simply for saving data for future examination.

9) There is a control button labelled [Plots] that allows you to define line graphs through the data. These plot line graphs are updated as the solution progresses (Top left). There will be a single Plot window available to view that represents the monitor line that was created in the geometry specification.

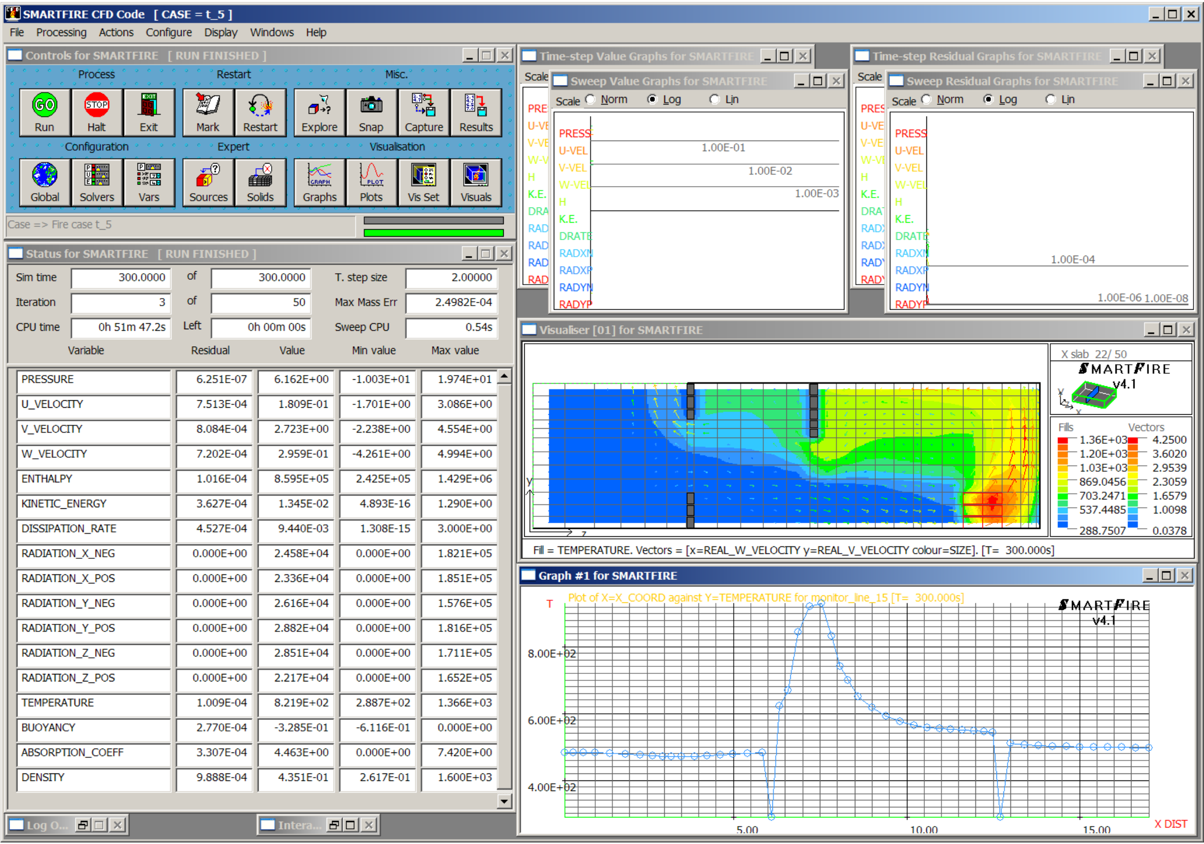

Figure 25-73 *SMARTFIRE* User Interface showing the end stage of the simulation, showing the fire plane visual and the monitored temperature plot graph.

25.6.6 STEP 5: INTERPRETING RESULTS FROM SMARTFIRE

If the simulation case has been run with a fairly course mesh, then we cannot totally rely on the results for any safety critical assessment of the modelled scenario. Also we may not have been very exacting about ensuring complete convergence (although a brief examination of the end of time step convergence history - the Time Step Residual Graph - indicates that most of the time steps ended at a reasonable state of convergence for this simulation). However we can use the results or data from the simulation to give a better understanding of the behaviour that we are likely to meet from this scenario.

The purpose of the simulation was to get a rough indication of the times before the rooms, neighbouring the fire, experienced conditions that are threatening to life and to compare this with the results from Tutorial 4, which did not have an extractor Fan. We are only considering the effect of temperature in the modelled scenario and effects such as smoke concentration, thermal radiation or the toxic gas concentration, are not considered. Again we can obtain a rough understanding of the fire effects by considering a typical harmful exposure time for a normal person to be approximately 1 minute at an air temperature of 180^{o}C. Remember that the SMARTFIRE CFD Engine is calculating temperatures in Kelvin. If we assume that, for safety considerations, reaching a critical temperature of 120^{o}C indicates the onset of dangerous - to human life - conditions. We can check through the plot graph data for the monitor line to determine at what time during the simulation each room experienced temperatures of 120^{o}C (i.e. 393K) or more in the vicinity of the monitor line.

Simple examination of the monitor line plot graph at the end of the simulation indicates that, even at the end of the simulation, the large room has not exceeded the critical temperature. Both of the monitored small rooms are experiencing temperatures in excess of 500K (i.e. 227^{o}C).

Further analysis of the saved graph plot data files reveals the following times to reach the critical temperature.

Monitored Room	Time until first monitored cell above $T_{critical}$	Time until all monitored cells above $T_{critical}$
Large Room (with single window)	134s	142s
Small Room nearest to fire (with Fan)	70s	102s
Small Room furthest from fire	120s	124s

Table 25-74 Table of times to detect critical temperatures in monitored rooms.

Unsurprisingly the extractor Fan has had an impact on the "safe" times for the monitored rooms, since a considerable amount of heat is being removed from the building via the small room that contains the Fan, whilst entrainment through the windows will have increased thus bringing more cool air into to the building. The large room experiences somewhat lower temperatures (for a longer period) than were experienced when no Fan was used. If the Fan extraction rate was increased, so the duration for safe occupancy would also be further increased.

It should be noted that this tutorial is not intended to give an exacting study into the fire safety considerations pertaining to a single fire modelling scenario but, rather, give an indication of the nature of possible simulations and the types of questions that can be asked of the simulation results.

25.6.7 STEP 6: EXITING THE CFD ENGINE

To exit the code, simply exit the CFD engine interface using the [Exit] button. On normal termination, the ***SMARTFIRE*** CFD engine will save a number of files that can be used for further visual post-processing (abnormal termination will not save any files and is encountered when the main window [X] button is pressed. Finally the CFD engine user interface will close and you will get back to the original geometry set-up tool. If you want to save any changes you have made to the fire modelling case, select the [File] item and then the [Save] option from the main menu.

It is interesting to note that the case directory (for a completed simulation) will contain many files that have been created during this exercise. Some of these files can be used for graphical post-processing and for re-starting this simulation from the stage when it was exited. Also there may be data capture files saved during your simulation.

You are advised to check the contents of the working case directories because the disk usage can be significant, particularly for large cell budget simulations.

This is the end of Tutorial 5.

25.7 *TUTORIAL 6*

25.7.1 OVERVIEW

This tutorial extends the scenario that was used in Tutorial 4 to use a gaseous combustion model for the fire and to use the more accurate multiple ray radiation model instead of the default six-flux radiation model. All of the other geometry and scenario specifications are identical to those in Tutorial 4 except for the changes required to the fire source to apply a mass release rate of a gaseous fuel.

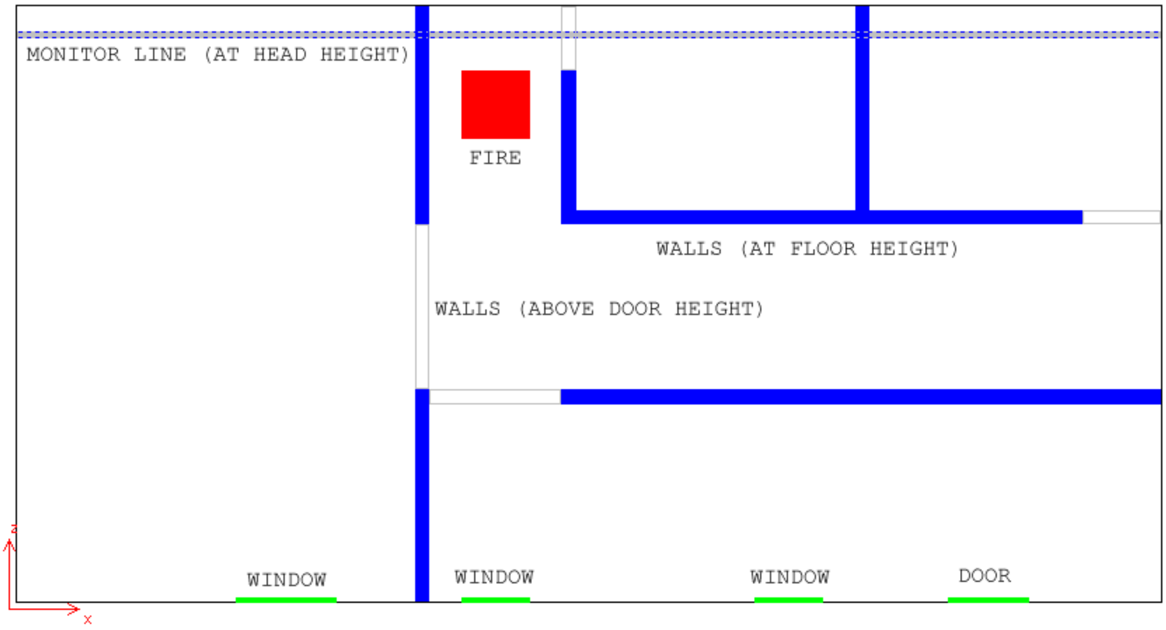

Figure 25-75 Plan view drawing showing the room layout geometry.

As with the Tutorial 4 scenario, certain assumptions have been made to simplify the specification as follows:

- The rooms are all assumed to be open plan during the fire simulation so all of the doorways and windows are fully open for all of the simulation.
- The doors on the end of the corridor are assumed closed throughout the simulation and they are of sufficient strength that the fire will not affect their structural integrity. There are actually windows present on the large outer wall of the largest room but these are assumed closed (and are hence not modelled) during the period of the simulation (in practice this cuts down the requirement for an additional extended region and hence makes the simulation run faster).
- Small air leaks around closed doors or windows are ignored and the heat transfer through the walls is considered to be negligible compared to heat transfer due to other processes, in the duration of the simulation.

- The corridor is assumed to have no combustible materials, which can support fire spread or secondary ignition.

The geometry of this case could be constructed from scratch but it is recommended that the user actually loads the ***SMARTFIRE*** model file created for Tutorial 4, and immediately re-saves it as a newly named case and then modifies the problem settings and the fire properties.

Again the fire is assumed to be a cleaning trolley fire at one end of the corridor. The fire will be specified as an equivalent methane fire that reaches a fuel release rate that is equivalent to the peak heat release rate of 2MW in two minutes, used in Tutorial 4. Since the fire is in the corridor and will prevent conventional means of egress from the building, we are actually interested in determining the times to reach a critical temperature (at about head height) in the neighbouring rooms where there are no exterior doorways to use for exit. This data will be compared to the results to Tutorial 4 to investigate the effects of using combustion and the multiple ray radiation models.

25.7.2 STEP 1: LOADING THE BASE CASE AND RENAMING

Run the ***SMARTFIRE*** case specification tool. Once the graphical user interface has opened, use [File] and [Open] to open the file loading menu. Use the file loading dialog window to browse to the folder containing the tutorial case from Tutorial 4 (probably called "t_4" or “tutorial_4”), typically found in the "smartfire\work" folder. Select the SMARTFIRE model file for this case (i.e. named "t_4.smf" - if using the naming convention suggested in Tutorial 4), and select the [Open] button. The base case from Tutorial 4 will be loaded and the default 3D wire frame view of the case will be displayed.

In order to prevent any subsequent scenario modifications from being accidentally written back into the Tutorial 4 case, it is recommended that the case is immediately renamed by selecting the [File] and [Save As] option. Use the file save dialog to browse back up to the "smartfire\work" folder - since you will currently be INSIDE the "t_4" folder, and enter the name "t_6" in the File name field and select the [Save] button. The case will be renamed and a newly named "t_6.smf" model file will be saved in a new "t_6" folder.

Now you can proceed to modify the settings to activate the combustion and multiple ray radiation models.

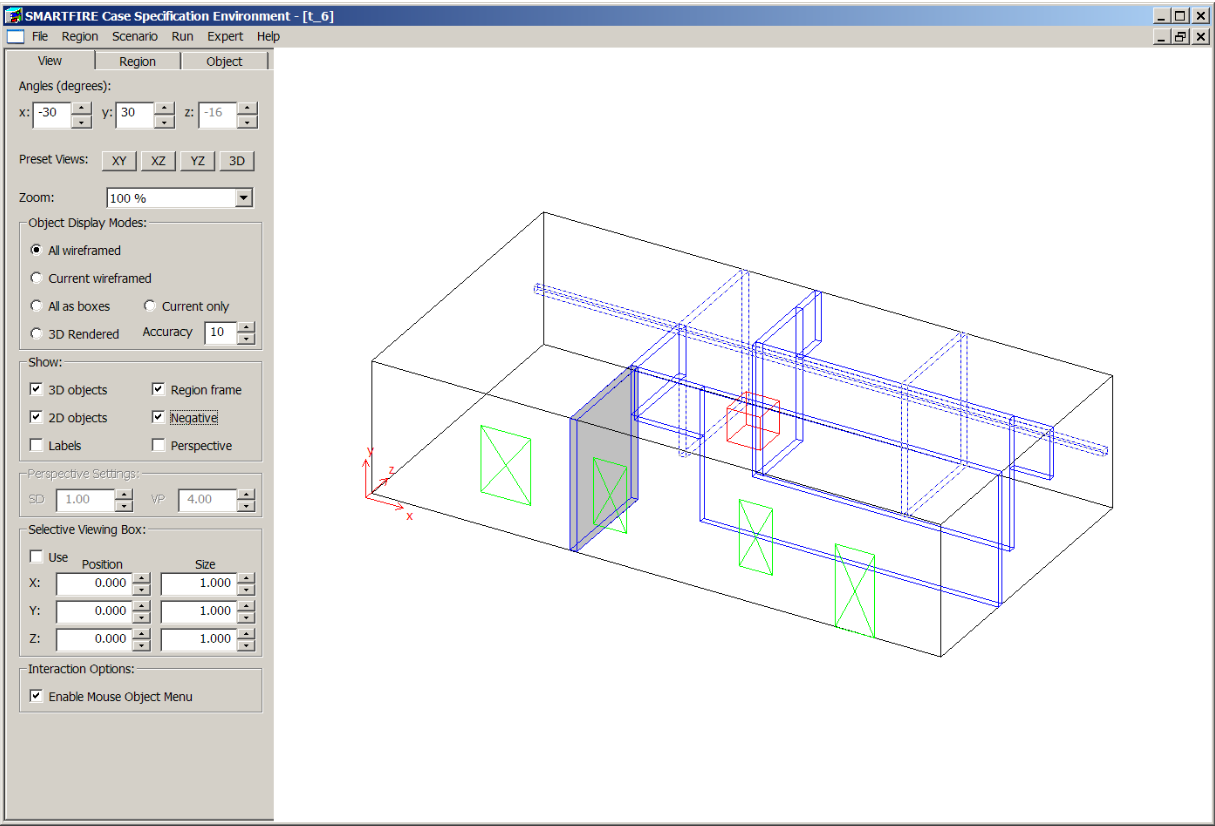

Figure 25-76 Specification tool showing renamed case from Tutorial 4.

25.7.3 STEP 2: ACTIVATING THE COMBUSTION MODEL

Select the [Scenario] option from the main menu and select the [Problem Type] settings. In the Problem Type Options window activate the combustion model by ticking the check box for the "Combustion Model".

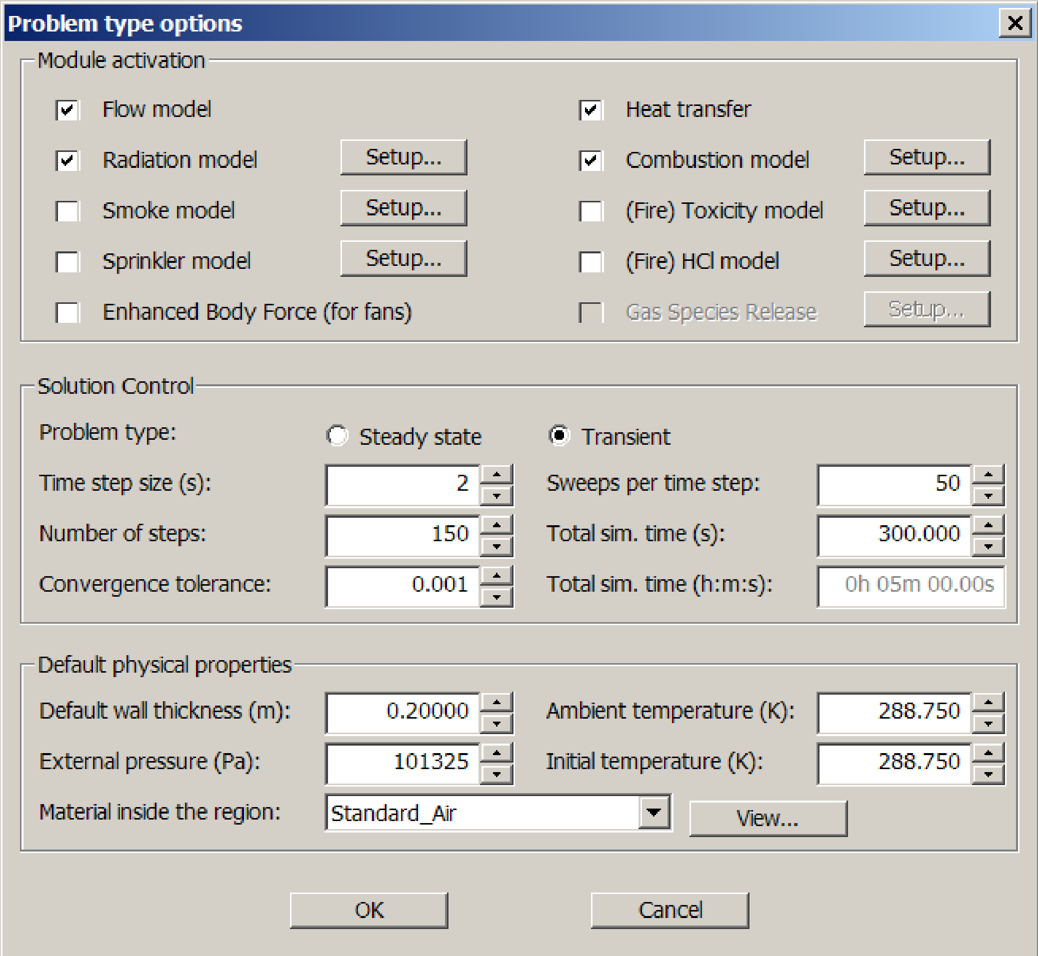

Figure 25-77 Problem Type options showing the activation of the combustion model.

All of the other Problem type options can be left to the settings used in Tutorial 4. I.e. Transient simulation using 150 time steps of 2.0 seconds per time step with 50 sweeps per time step. The convergence tolerance is set to 0.001. The default wall thickness is 0.2 m. The Ambient temperature and the Initial temperature are set to 288.75K. The external pressure is 101325 Pa and the building is set to contain "Standard_Air".

Once the Combustion model has been activated, enter the combustion model settings using the [Setup...] button near the combustion model check box. This will open a window that shows the default settings for the combustion model. The default combustion fuel is for a methane fire, so none of the parameters should need to be changed BUT it is always advisable to check that the combustion parameters are set to the correct defaults since these settings are being loaded from a previous case that could have been modified.

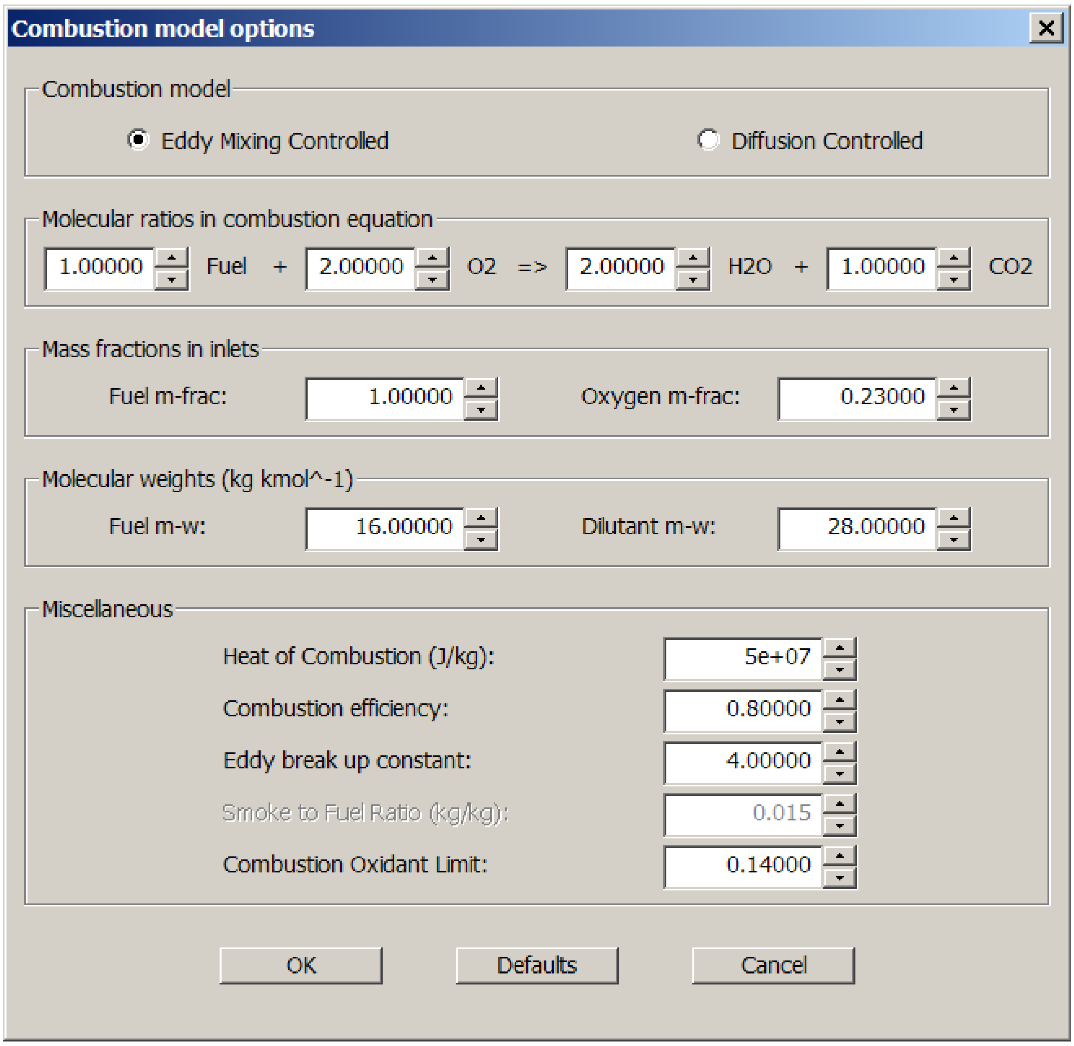

Figure 25-78 Combustion model options.

The combustion model should use the Eddy Mixing Controlled combustion. This is the more sophisticated combustion model since it links the combustion to the turbulence model.

The molecular ratios in the combustion equation are correct for methane when the following parameters are used:

$\underline{\mathbf{1}}\ CH_4 + \underline{\mathbf{2}}\ O_2 \Rightarrow \underline{\mathbf{2}}\ H_2O + \underline{\mathbf{1}}\ CO_2$

The fuel mass fraction in inlet is left as the default value of 1.0 (i.e. pure fuel inlet) and the oxygen mass fraction in inlet is set to the default of 0.23 (i.e. the normal ratio of oxygen in air).

The Molecular weights are set to 16.0 for the Fuel (i.e. Methane = CH_4 = Carbon$^{(a.w.=12)}$ + 4 x Hydrogen$^{(a.w.=1)}$ = 12 + 4 = 16) and 28.0 for the dilutant in Air (i.e. Nitrogen = N_2 = 2 x Nitogen$^{(a.w.=14)}$ = 28). N.B. the $^{a.w.}$ number represents the Atomic Weight of the atoms of the specified elements and is used to calculate the Molecular Weight. Typically the average Atomic Weight should be used taking into account the relative abundance of different isotopes of the element.

The Heat of Combustion for Methane in Air is 5.0e7 J/kg. The Combustion Efficiency is taken as the default 80% (i.e. 0.8) and the Eddy Break up constant is left as the default value of 4.0. The combustion oxidant limit specifies the fraction of oxidant in the atmosphere required to sustain combustion. This factor will only play a part for very fuel rich environments, limited ventilation environments or, possibly at the heart of a large fuel source volume. The default value equates to 14% (i.e. 0.14) of oxygen needed to sustain combustion.

Now it is necessary to create an equivalent fire source for the release of gaseous fuel. The parameters that were used for the Heat Release Simple Fire were for an initial 50 kW raising to 2 MW after 120.0 seconds. The time equation parameters were A=50.0, B=0.0, C=0.1285, D=0.0 and E=0.0 with Start time=0.0 and End time=120.0 seconds. The conversion is achieved by dividing the Heat release rate terms (of the volumetric heat release fire) by the Heat of combustion for the particular fuel being used. In this case we have A=150.0e3/5.0e7=3.0e-3 and C=0.1285e3/5.0e7=2.57e-6 (Remember that the terms were in kW but we need to do the calculation in Watts and Joules to give kg of fuel release rate). Entering these values into the Fire Properties window for the fuel release rate we obtain the following fuel-equivalent fire:

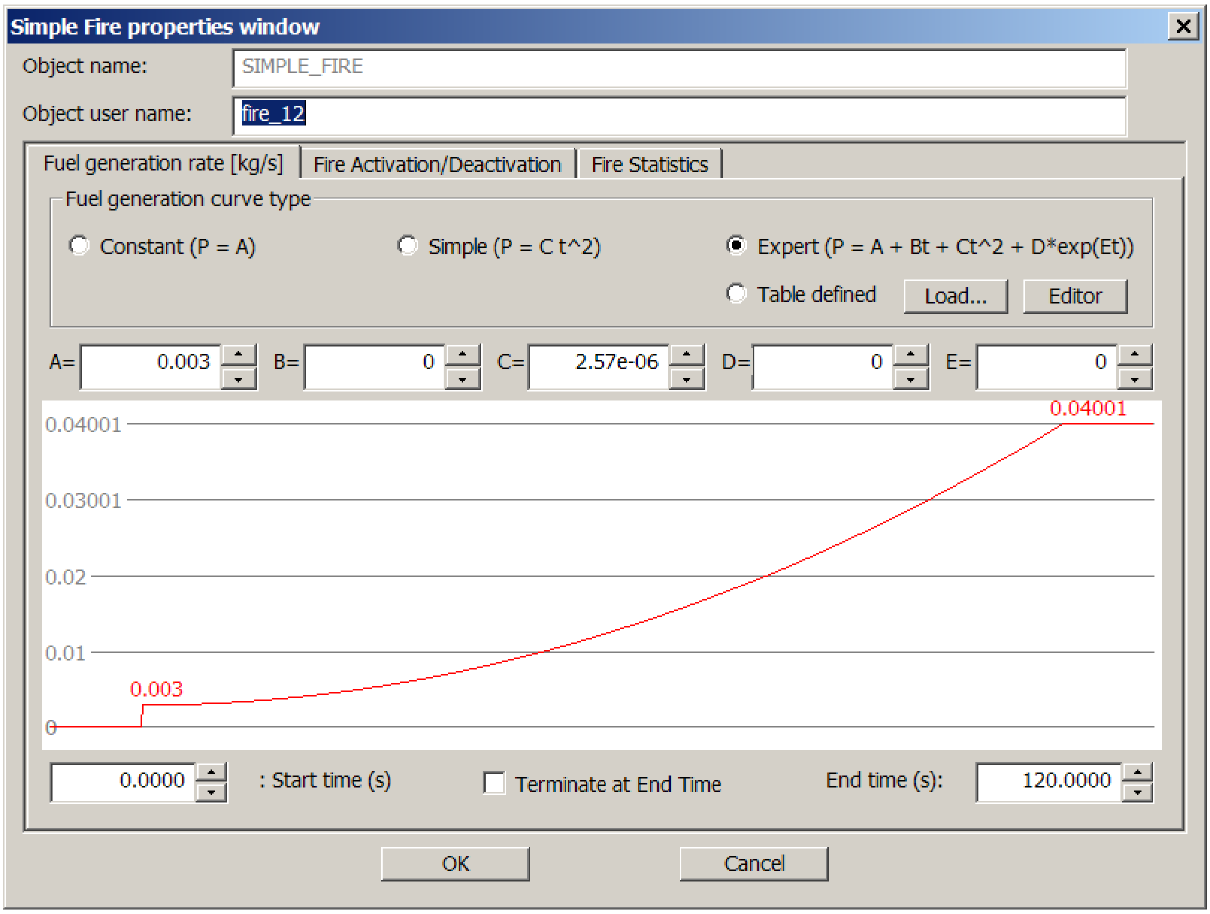

Figure 25-79 Fire properties for the equivalent fuel release rate.

25.7.4 STEP 3: ACTIVATING THE MULTIPLE RAY RADIATION MODEL

Select the [Expert] option from the main menu and select the [Expert Options] item. This opens a simple dialog where there are two check boxes. The User must check the [Enable

Expert problem set up] as this unlocks some of the additional functionality, such as the Multiple Ray radiation model. This is necessary because the Multiple Ray Radiation model should not be used by novices on arbitrary cases since it is possible (with poor choice of rays or by particular geometrical alignments of fires and free surfaces, to decouple the fire from the free surfaces and thus make the multiple ray radiation model perform badly. Select the [Scenario] menu option and the [Problem type] item to access the Problem type options window. The radiation model should already be active from the previous tutorial. Select the [Setup…] option (near the Radiation check box) to enter the Radiation model options window. The multiple ray radiation model option is now available and can be selected using the appropriate radio button from the Radiation Model options window.

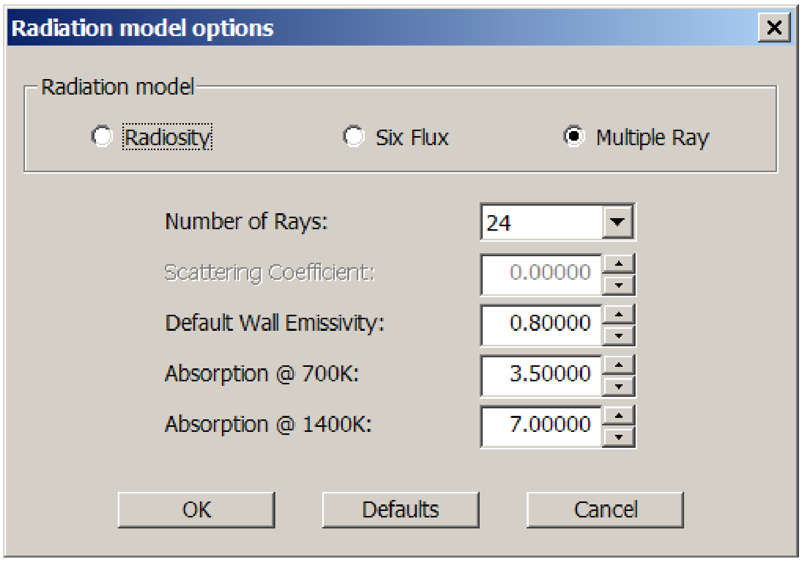

Figure 25-80 Radiation model options window showing the selection of Multiple ray radiation using 24 rays.

In this tutorial it is recommended that 24 rays are used to model the Multiple Ray radiation.

25.7.5 STEP 4: CREATING A MESH FOR THE SIMULATION

Once you are satisfied with the modifications to the physics handling of the case you need to run the mesh creation system. To run the automated meshing tool from the main menu, choose the [Run] option and the select [Create Mesh]. This will open the mesh creation tool user interface. The automated meshing tool will first determine if the current geometry already has an existing mesh previously created. If one is available then the user will be given the option to load the existing mesh or to create a new one. Since we have not actually changed the geometry from Tutorial 4, then it is permissible to use the existing meshing, if this is available. If an existing mesh is not available then it is quite a simple task to re-specify the mesh using the following instructions.

Once all of the geometry and modelling issues have been resolved, the meshing system will analyse the geometry and present a Cell budget window. This has the automated meshing system's recommended cell budget with individual user-selectable cell budgets for each of the co-ordinate directions. It is recommended that this case be meshed as a course mesh (usually a good option for checking that a case has been correctly specified and that all of the physics and numerics behave as expected). Unfortunately, for a course mesh, the meshing system will struggle to create a very even mesh. Hence we will use the manual mesh editor to selectively refine certain mesh blocks (mesh slices between and through objects) to give a reasonably even course mesh. The Cell Budget window appears as follows:

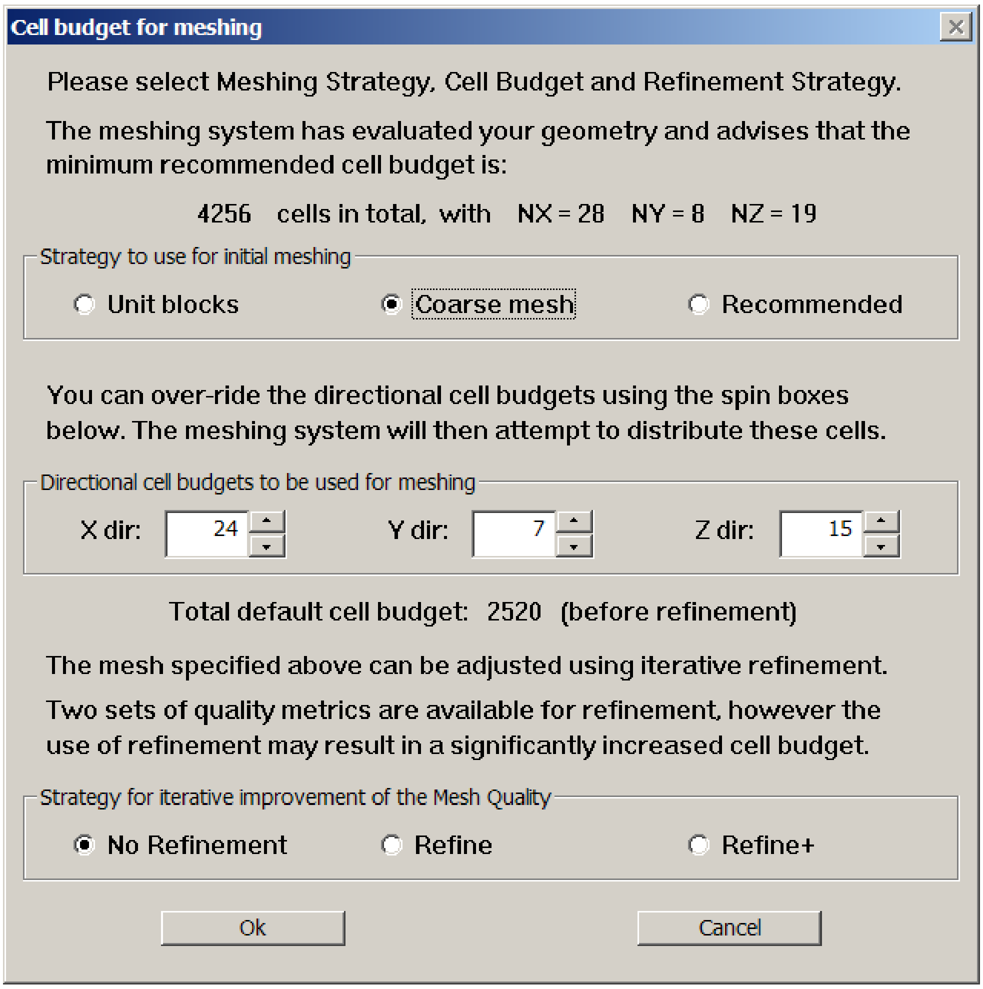

Figure 25-81 Cell budget for meshing.

The first attempt at meshing using a course mesh appears as follows:

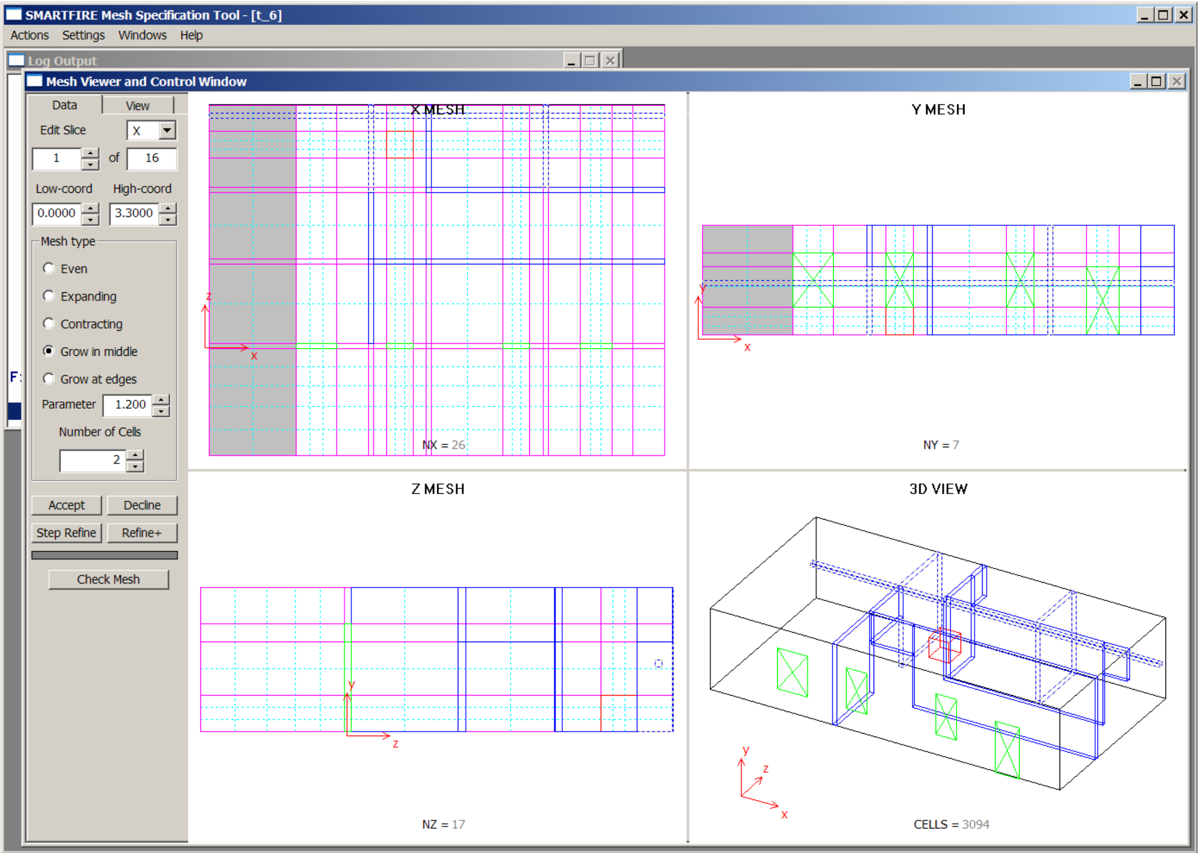

Figure 25-82 First course mesh from the automated meshing system.

This course mesh would only be appropriate for a quick test run of the simulation to check that the configuration is working as expected. For a production run, where accurate results are required, the user is recommended to use the [Refine+] button to ensure that the mesh is of sufficient quality.

It would also be possible to manually refine the mesh to create a suitably even mesh by adjusting the numbers of cells in the mesh slices. It should be noted that the "slice" selection combo box and slice number spin box can be used to highlight a slice for refinement but it is generally easier to point at the required slice in the display windows and select (with the mouse pointer and button) to make the current slice active (highlighted with grey=internal region or green=external region shading). Each of the mesh display windows allows either of the displayed directions to be edited. The current block to be edited will be highlighted and the [Data] panel will allow the number of cells in that slice to be changed.

There is no absolutely correct mesh for any simulation and, with experience, the user will eventually develop knowledge of when and where cells need to be refined. Generally the following points should be considered when manually refining a mesh.

(1) The fire will be generating a highly buoyant plume that may lean over as air is entrained or channelled around obstacles. It is good practice to provide additional cells in the vicinity of the fire to give the plume as much freedom as possible.

(2) Vents are likely to have higher speed channelled flows and will consequently benefit from having as many cells as possible across their height and width.

(3) Where a buoyant plume impacts a ceiling (or wall) there should be as many cells as possible to allow for plume rotation into ceiling jets.

(4) It is not a good idea to have largely dissimilar sizes between adjacent mesh cells.

(5) It is not a good idea to have very squashed (or elongated) mesh cells.

The result of using the [Refine+] option on the mesh will give the following mesh:

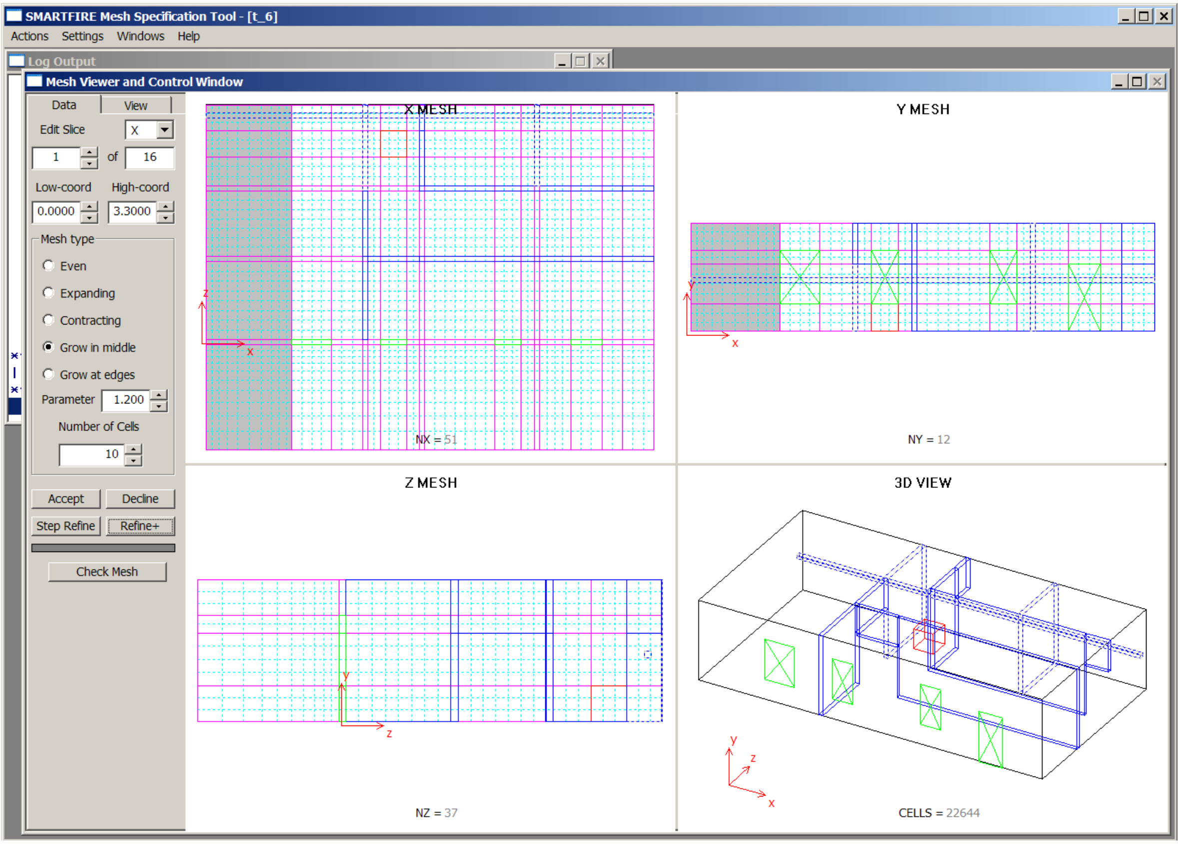

Figure 25-83 Manually edited and refined mesh.

The mesh is now ready and can be saved for simulation. Pressing the [Accept] button will allow the mesh and scenario specification to be saved as a set of specification files for the CFD Engine to read. Pressing the [Accept] button will also cause the automated meshing tool to close and control will then return to the ***SMARTFIRE*** Case Specification Environment.

25.7.6 STEP 5: RUNNING THE CFD ENGINE

The final stage involves performing the numerical simulation itself using the CFD engine component of ***SMARTFIRE***. To run the CFD engine select on the main menu item [Run] and the [Run CFD Engine] option. This will launch the numerical CFD engine that will automatically load the case specification and mesh that you have just created. You may have to be a little patient as this stage involves a considerable amount of file parsing, memory allocation and initialisation. Eventually the user interface for the CFD engine will appear completely. It should be noted that if you are using a low resolution display then the CFD engine use-interface may display itself with more compact windows and a slightly different layout (e.g. with more of the windows initially closed) in order to create a usable display.

Once the CFD Engine has loaded the data and performed all of the require initialisations then the User Interface will be displayed with the un-processed view of the simulation case. The initial CFD Engine display will appear as follows:

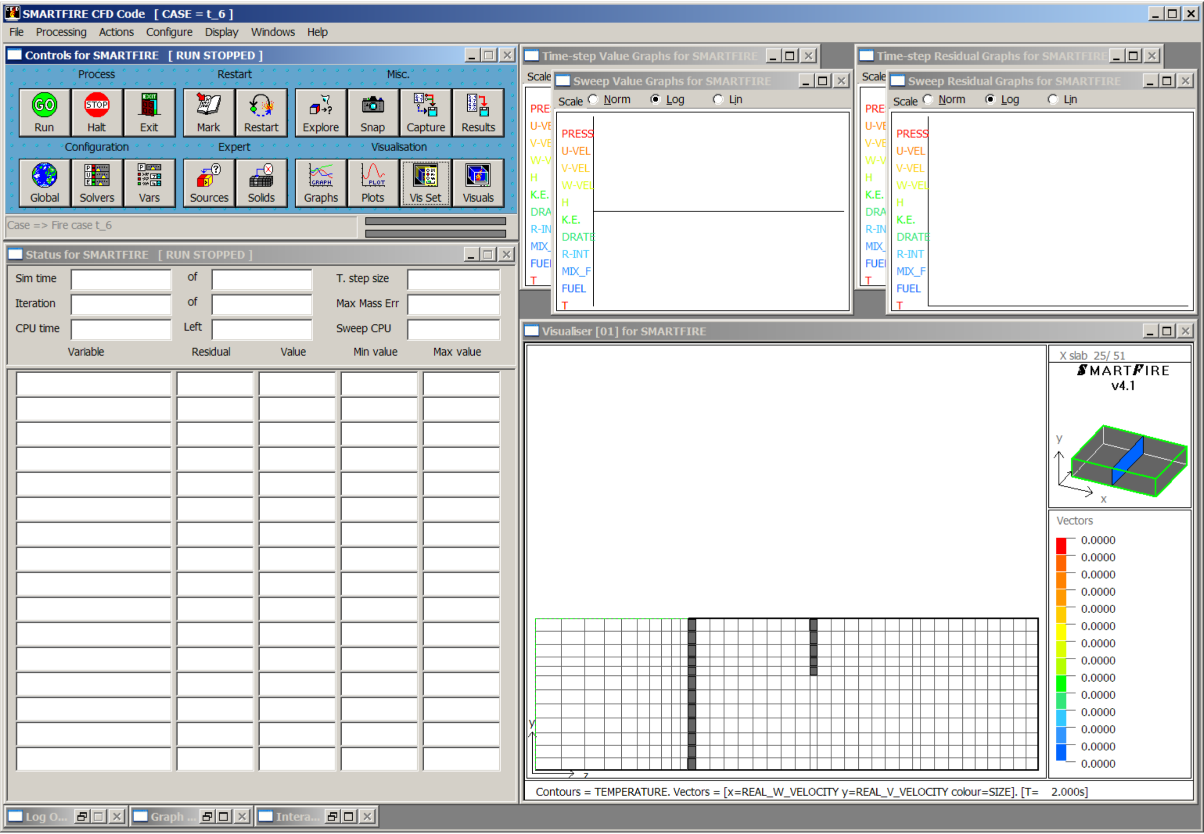

Figure 25-84 *SMARTFIRE* CFD Engine user interface at start up.

The Case Specification Environment will have selected a default view plane for the visualisation (typically at the halfway layer of cells in the X-direction). Unfortunately this view plane might not be a particularly interesting one to display, so it is worth selecting a good plane to view throughout the simulation. In this case we have not kept any information about what would be a good plane to select but we can use the visual configuration window to "find" a suitable plane. If we press the [Visual] button then the Visual Configuration Window will open and we can use it to investigate the arrangement of the objects so that a more interesting slice plane can be found. By selecting the lowest Y slice, we can see where the fire is located in the Visual Display window.

It is then a simple matter of re-entering the Visual Configuration Window and selecting the planar slice that will give a more meaningful display of what is occurring in the vicinity of the fire.

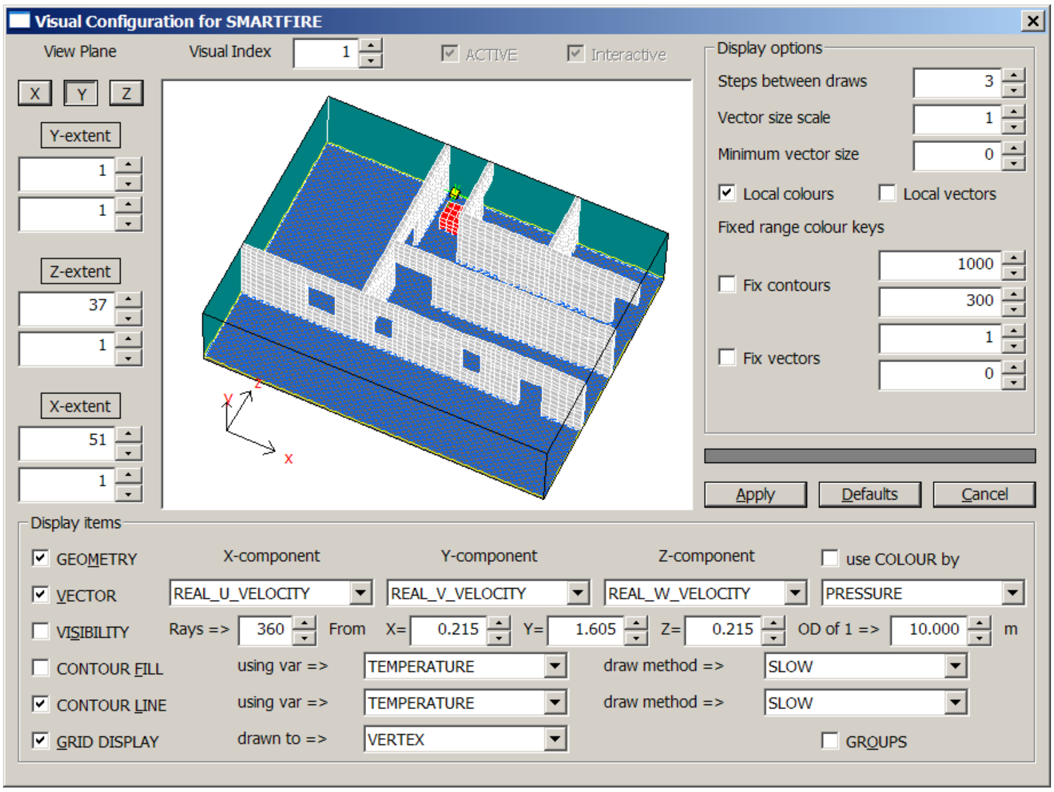

Figure 25-85 Visual Configuration selecting the lowest Y-layer of cells to view.

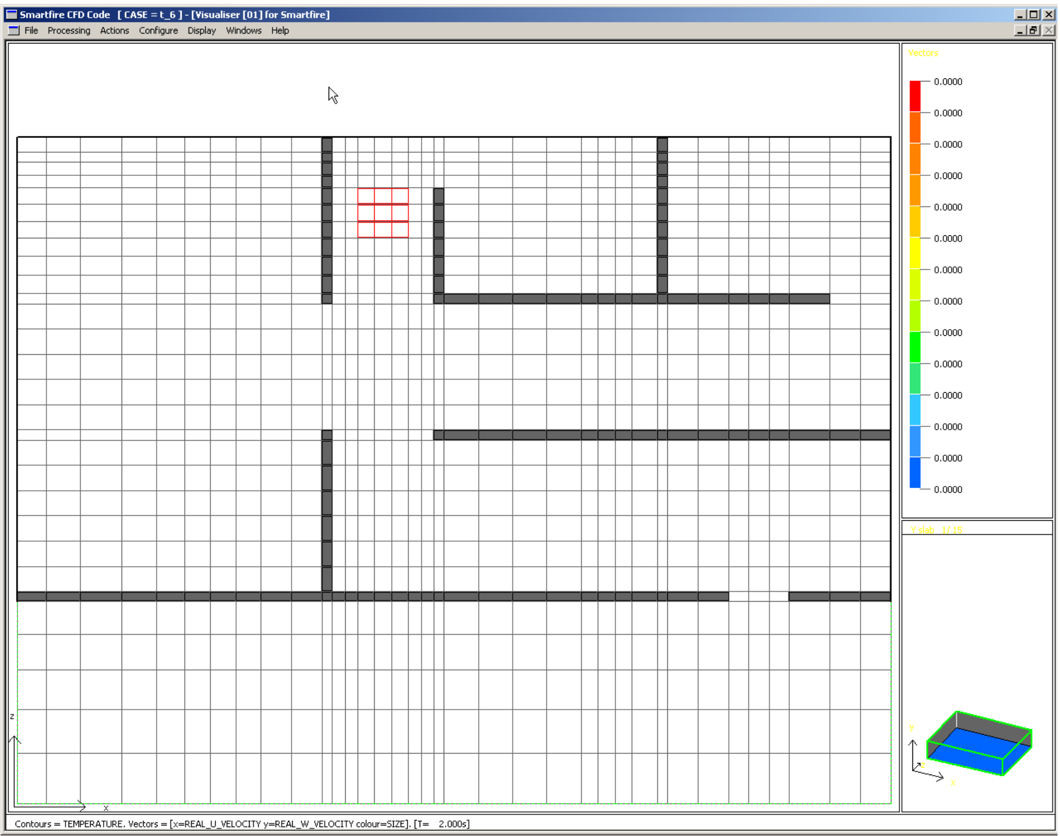

Figure 25-86 Visualisation of the lowest Y-slice of cells showing the fire location.

After selecting the X=22 slice (the exact index will depend on the mesh used but the user should check that it is through the centre of the fire) the following Visual Display will appear on the User Interface.

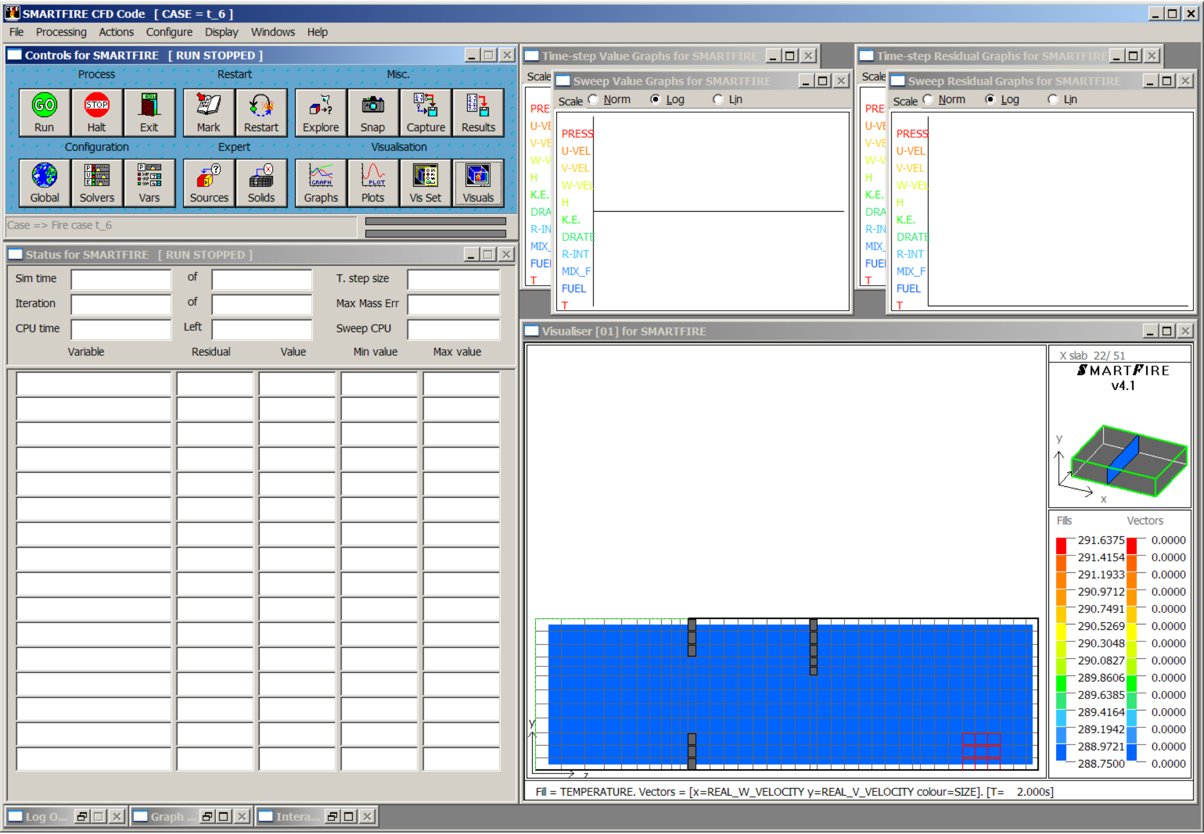

Figure 25-87 *SMARTFIRE* User Interface after selection of a more interesting visual slice plane through the fire.

It should be noted that the Visual Configuration Window could also be used to change the features that are displayed in the data Visualisation. In the image above the [Contour Fill] option of "TEMPERATURE" has been activated because this gives a clearer representation of the temperatures in the fire plane, especially when using vectors of velocity in the same display.

Before starting the simulation it is worth considering the forms and frequency of data capture from the simulation. If the user merely wishes to see what conditions are like at a particular time then the simulation can simply be run until that required simulation time is reached. In our scenario we are not sure what time the conditions will occur so we need to save sufficient data to allow us to analyse at what time certain conditions were experienced. The user should open the [Configure] menu from the main menu and select [Data Capture]. This presents the Data Capture Configuration menu, where the frequency and nature of the data that will be saved during (and at the end of) the simulation, can be configured. In this simulation it is recommended that the user save the results, graphs and visuals at the end of every time step. This is configured by ticking the appropriate check boxes in the "Automatic Transient Outputs" section of the menu.

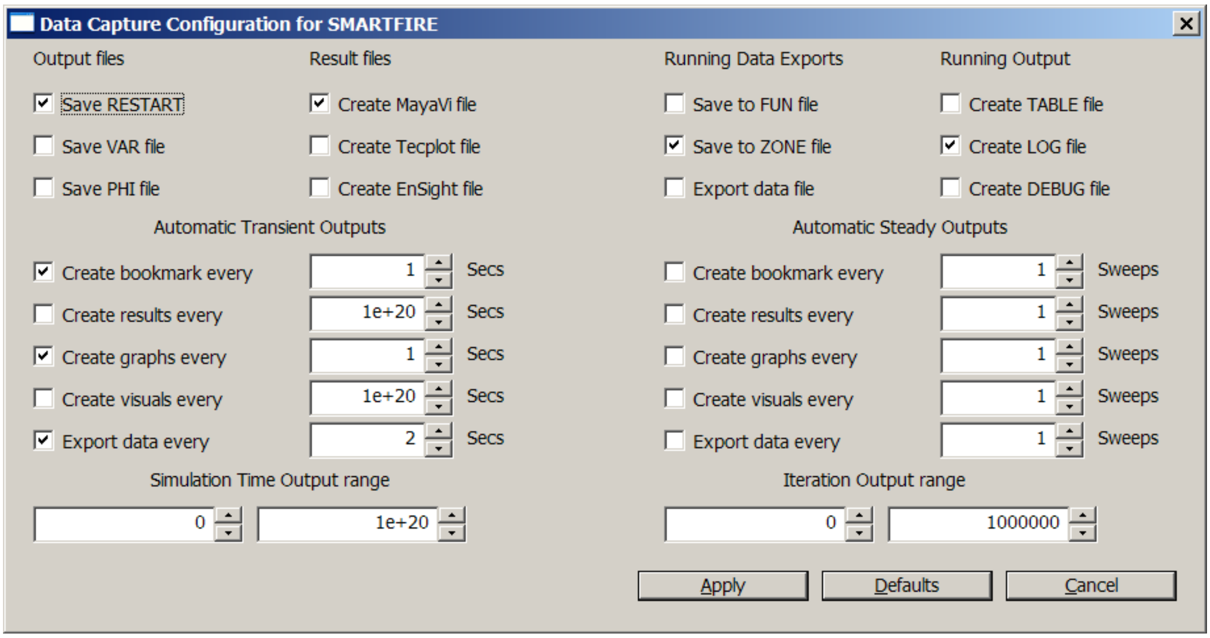

Figure 25-88 Data Capture Configuration menu.

Now that everything has been configured it is possible to start the processing of the simulation scenario.

In order to start the numerical simulation, you will need to press the [Run] button, marked with a GREEN (GO) icon. This will start the simulation process. To halt the simulation at any time, press the [Halt] button marked by a RED (STOP) icon. You can also terminate the simulation at any point by pressing the [Exit] button (showing a little door icon). There are a number of extra buttons and other controls that allow experts to intimately control the configuration of the solution process. However, non-experts are not recommended to make any changes to the control settings unless this is recommended in the user guide.

The graphical windows of the user interface present various views of the data and the status of the simulation during the simulation process. The things to look for are:

1) The residual graphs that are indicators of the convergence of the solved and calculated variables of the numerical simulation process (Top right).

2) The monitor values and variable residuals (current solution error states) of various important variables are shown in the status window (Bottom left).

3) The emerging vector flow and temperature contour patterns for the particular selected slice of the room in the Visualisation window (Bottom right). ***SMARTFIRE*** is now able to maintain multiple visualization windows, each containing completely independent view and display selections.

4) The data ranges for each variable displayed in the status window (Bottom left).

5) The control window has progress bars in its bottom right corner (below the visualisation buttons). These bars indicate solution progress. The upper bar is filled once every sweep whilst the lower bar is filled once for the whole configured

simulation (Top left).

6) The status window has displays indicating the sweep number and time step number (only for transient simulations) to indicate the current stage of processing (Bottom left).

7) The status window has estimates for the CPU time taken and remaining. These are only estimates but can give a reasonable approximation of the expected duration of a simulation (Bottom left).

8) A key feature of ***SMARTFIRE*** is the access to save a bookmark and restart from saved bookmarks at any time. The control button labelled [Mark] will drop a bookmark of the current stage of the solution into a database for this case. The button labelled [Restart] allows a previous bookmark state to be loaded as if subsequent processing had not happened. This can be invaluable for problematic simulations that need expert solution control or simply for saving data for future examination (Top left).

9) There is a control button labelled [Plots] that allows you to define line graphs through the data. These plot line graphs are updated as the solution progresses (Top left). There will be a single Plot window available to view that represents the monitor line that was created in the geometry specification.

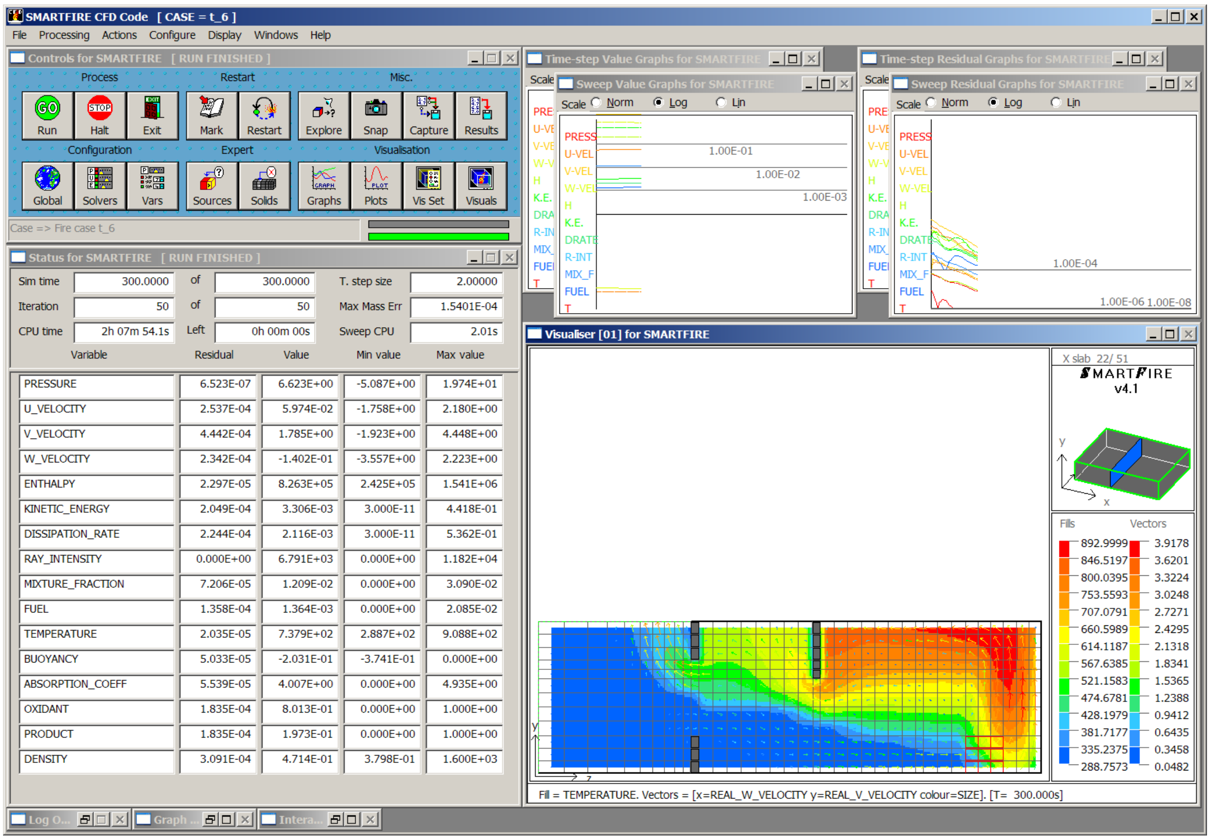

Figure 25-89 *SMARTFIRE* User Interface showing the end stage of the simulation.

25.7.7 STEP 6: INTERPRETING RESULTS FROM SMARTFIRE

As with Tutorial 4, if the simulation has been run with a course mesh then we cannot totally rely on the results for any safety critical assessment. Again, we have not been very exacting about always ensuring complete convergence (although examination of the end of time step convergence history - the Time Step Residual Graph - indicates that most of the time steps ended at a reasonable state of convergence for this course mesh simulation).

The purpose of the simulation was to get a rough indication of the times before the rooms, neighbouring the fire, experienced conditions that are threatening to life. We are only considering the effect of temperature in the modelled scenario and effects such as smoke concentration, thermal radiation or the toxic gas concentration, are not considered. Again we can obtain a rough understanding of the fire effects by considering a typical harmful exposure time for a normal person to be approximately 1 minute at an air temperature of 180^oC. Remember that the SMARTFIRE CFD Engine is calculating temperatures in Kelvin. If we assume that, for safety considerations, reaching a critical temperature of 120^oC indicates the onset of dangerous - to human life - conditions. We can check through the plot graph data for the monitor line to determine at what time during the simulation each room experienced temperatures of 120^oC (i.e. 393K) or more in the vicinity of the monitor line.

Simple examination of the monitor line plot graph at the end of the simulation indicates that all of the monitored rooms are experiencing temperatures in excess of 410K (i.e. 137^oC). This is lower than the final temperatures found in Tutorial 4, by some 90^oC.

In this case we are also interested in seeing what effect using the multiple ray radiation model and the combustion model have had on the results that were found in Tutorial 4. As a reminder the results found from the Tutorial 4 simulation were as follows:

Monitored Room	Time until first monitored cell above $T_{critical}$	Time until all monitored cells above $T_{critical}$
Large Room (with single window)	118s	126s
Small Room nearest to fire	78s	106s
Small Room furthest from fire	104s	116s

Table 25-90 Table of times to detect critical temperatures in monitored rooms from Tutorial 4 (using volumetric heat release and six flux radiation).

Further analysis of the saved graph plot data files for Tutorial 6, reveals the following times to reach the critical temperature.

Monitored Room	Time until first monitored cell above $T_{critical}$	Time until all monitored cells above $T_{critical}$
Large Room (with single window)	144s	156s
Small Room nearest to fire	126s	134s
Small Room furthest from fire	126s	140s

Table 25-91 Table of times to detect critical temperatures in monitored rooms from Tutorial 6 (using combustion and multiple ray radiation using 24 rays).

Now we see that the use of the combustion model and the multiple ray radiation model have apparently extended the safe times for the monitored rooms by times between approximately a half a minute (for the large room and the small room remote from the fire) to a minute (for the small room nearest the fire). The large variation in times is to be expected. The use of the combustion model and the multiple ray radiation model will, generally, give better approximations to the real conditions in the fire scenario, at the expense of considerable extra computation. In the Tutorial 4 simulation the CPU time used was 4h 0m and 27s, and for Tutorial 6, the CPU time used was 9h 3m and 16s, indicating that the combustion model and the multiple ray radiation model extend the CPU time used by at least double. It is also pertinent to ask if this accuracy is necessary for the particular usage of the results. In the analysis above we are looking for a safe time for room occupancy and it is probably acceptable to have a conservative estimate of this time, as we would by having slightly over estimated temperatures - typical of the volumetric heat release model for the fire.

It should be noted that this tutorial is not intended to give an exacting study into the fire safety considerations pertaining to a single fire-modelling scenario but, rather, give an indication of the nature of possible simulations and the types of questions that can be asked of the simulation results.

To reiterate, the decision about which modelling techniques and settings to use, can have a large impact on the results obtained from the simulations. It should be noted, however, that it is not always necessary to opt for the best available modelling techniques due to the, possibly unacceptable, protracted run-times or the fact that a known over- or under- estimate will give a conservative safety margin.

25.7.8 STEP 7: EXITING THE CFD ENGINE

To exit the code, simply select the [Exit] button on the user interface. On normal termination, the ***SMARTFIRE*** CFD engine will save a number of files that can be used for further visual post-processing (abnormal termination will not save any files and is encountered when a critical error occurs or when the main window [X] button is pressed). Finally the CFD engine user interface will close and you will get back to the original geometry set-up tool. If you want to save any changes you have made to the fire modelling case, select the [File] item and then the [Save] option from the main menu.

It should be noted that the case directory (for a completed simulation) will contain many files that have been created during this tutorial. Some of these files can be used for graphical post-processing and for re-starting this simulation from the stage when it was exited. Also there may be data capture files saved during your simulation. Users should be aware that running many large simulations can rapidly fill up hard disks with results data.

This is the end of tutorial 6.

26 *SMARTFIRE* ADVANCED TUTORIALS

26.1 *INTRODUCTION*

The following tutorials are intended for you to work through various aspects of building, running and analysing a CFD based fire simulation using ***SMARTFIRE***.

The first four cases involve a small scale geometry with two rooms connected by a corridor. One of the rooms is vented to the outside. These four cases explore increasing sophistication in the fire scenarios and allow the user to experience various capabilities of the tools.

- Tutorial A1 involves setting a simple heat release rate fire source in one of the rooms with all connecting doors open.
- Tutorial A2 uses the post processing and data analysis facilities of the ***SMARTFIRE*** CFD Engine to analyse the Tutorial A1 case.
- Tutorial A3 uses output results from Tutorial A1 to investigate the post-processing and animation facilities of the DataView tool.
- Tutorial A4 uses gaseous combustion and involves setting a secondary fire source in the second room which ignites at a pre-set temperature. The connecting door is initially closed.
- Tutorial A5 introduces CAD file import and allows a scenario to be created from a large scale floor plan diagram.
- Tutorial A6 makes use of the parallel version of the ***SMARTFIRE*** CFD Engine to simulate the two room scenario.

In all cases, the user is advised to take a note of the level of convergence that is achieved for the mesh that is being used. These tutorials will mostly be using coarse meshes so that you will be able to see the outcome of the simulation in a reasonable time – but typically it is recommended that a production run would use much finer meshes (and consequently require proportionately longer processing times).

26.2 *TUTORIAL A1*

26.2.1 GEOMETRY SPECIFICATION

The geometry has three interconnected compartments as shown in the diagram below.

In this Tutorial all of the doors are assumed to be open throughout the simulation.

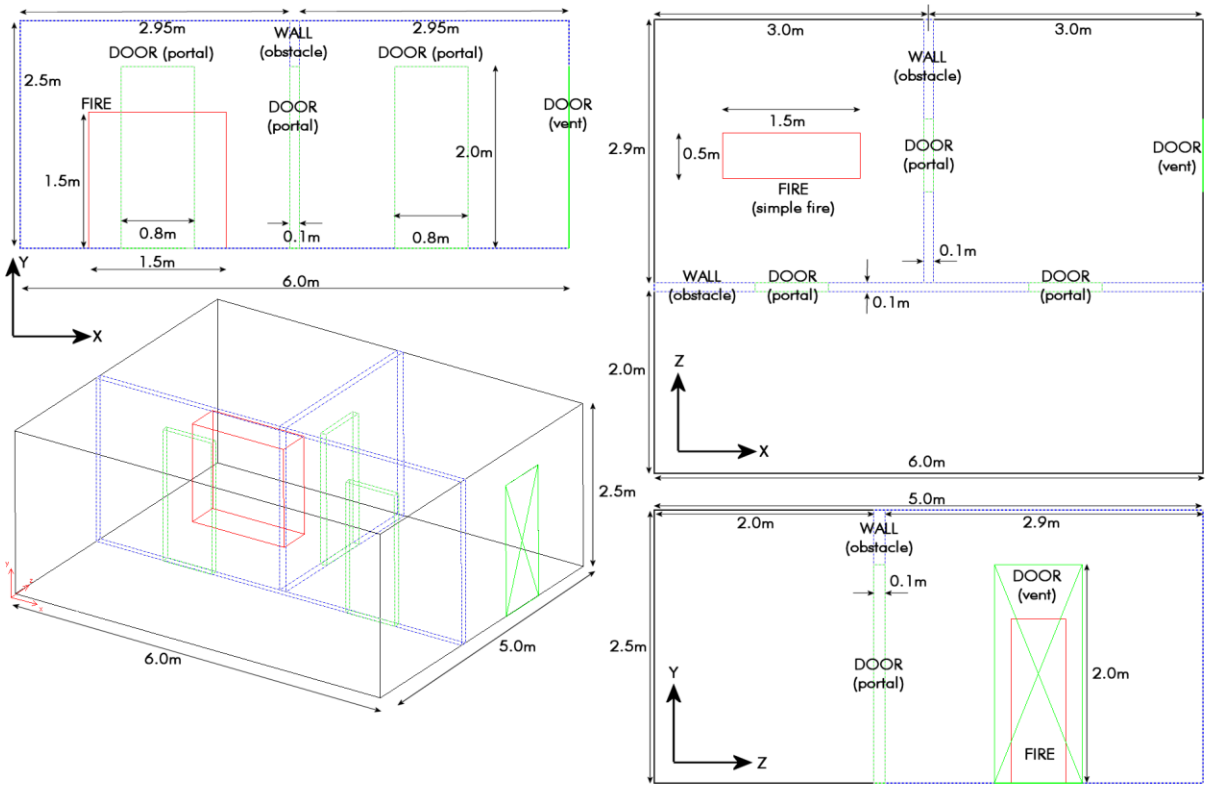

Figure 26-1 : Three room geometry with tall fire and open doors.

26.2.2 SCENARIO SPECIFICATION

<u>Size the geometry region:</u>

Region size is (6.0, 2.5, 5.0).

<u>Create the following objects</u> *(using the Case Specification Environment):*

Long wall [OBSTACLE]
at (<u>x=</u>0.0, <u>y=</u>0.0, <u>z=</u>2.0) size (<u>dx=</u>6.0, <u>dy=</u>2.5, <u>dz=</u>0.1).

Short wall [OBSTACLE]
at (2.95, 0.0, 2.1) size (0.1, 2.5, 2.9).

Open door [PORTAL open in Z]
at (1.1, 0.0, 2.0) size (0.8, 2.0, 0.1).

Open door [PORTAL open in Z]
at (4.1, 0.0, 2.0) size (0.8, 2.0, 0.1).

Open door [PORTAL open in X]
at (2.95, 0.0, 3.1) size (0.1, 2.0, 0.8).

Open door [VENT on high X surface]
at (6.0, 0.0, 3.1) size (0.0, 2.0, 0.8).

Fire [SIMPLE FIRE]
at (0.75, 0.0, 3.25) size (1.5, 1.5, 0.5).

<u>Select the Fire and Configure the Fire Properties as:</u>

Expert Heat release curve
with A=50, C=0.005 and END_T=300s (rising to 500kW).

<u>Configure the Scenario => Problem Type as:</u>

Flow, Heat and Six flux radiation,
Transient with 30 time steps of 10 seconds,
50 sweeps per time step. Select OK the accept the settings.

26.2.3 MONITORING

<u>Create the following Monitoring Objects:</u>

Create plot graphs [MONITOR LINE objects] for

1) TEMPERATURE distribution against Y-COORD.

and

2) horizontal velocity distribution against Y-COORD along the vertical centre line of each of the four doors.

3) TEMPERATURE distribution against Y-COORD for a vertical stack in the centre of the corridor.

Note that the horizontal velocity component will depend on the direction of the opening so an X facing opening will have the U-VELOCITY as the horizontal velocity component.

Save these distributions at 30 sec, 1, 2 and 3 minutes by setting
Scenario => Output Control => Use Transient Graph Save

Set the save interval to every 30 seconds (or every 3 time steps).

This will create files named as follows:
"plot_t_data_*#plot_num_@#time*.gpd"

26.2.4 DATA GATHERING

(IMPORTANT --- FOR TUTORIAL A3)

Configure result file saves at 10 sec intervals using
Scenario => Output Control => Use Transient Results Save

Set the save interval to saving at every time step (i.e. every 10 seconds).

26.2.5 MESH AND SIMULATE

<u>Meshing options:</u>

Save the case as "*tut_case_A1*".

Select Run => Create Mesh.

Choose to ignore overlaps (we want PORTALS to make openings through walls).

Select the geometry type as "Standard Room Geometry".

Stop the meshing system from adding an extra extended region in Y+ direction.

Choose Cell Budget for Meshing as "Recommended mesh" with the "Refine" option.

Accept the mesh (this will create script files for the ***SMARTFIRE*** CFD Engine).

<u>Run the simulation:</u>

Select the Run => Run CFD Engine.

Wait for the CFD Engine to initialise and open fully.

Press the [GO/Run] button to start the simulation.

Stop and examine data/settings at any time with the [Stop] button.

<u>DO NOT</u> exit or close the CFD engine at the end of the simulation as you will need SMARTFIRE in this end simulation state for TUTORIAL A2.

26.2.6 RESULTS AND ANALYSIS

<u>Results/Observations:</u>

What is the maximum temperature at the end of the 3 minute simulation?

Take a note of the level of convergence that has been achieved.

Do you think this represents an acceptable level of convergence?

Ensure that you have saved the monitored data, i.e. the temperature and horizontal velocity distribution through the vertical centre line of each of the doors and the vertical temperature distribution in the centre of the corridor at 30 sec, 1,2 and 3 minutes into the simulation.

You will need this data to compare with the other tutorial case.

26.3 *TUTORIAL A2*

26.3.1 RUTIME DATA ANALYSIS IN THE INTERACTIVE CFD ENGINE

In this Tutorial you will use some of the data analysis tools in the ***SMARTFIRE*** CFD Engine to post-process and analyse the data from your existing Tutorial Case A1 simulation. It is assumed that Tutorial Case A1 has now finished running and that the CFD Engine is waiting at the end of the simulation.

The analysis tools used are:

- The *Data Explorer*,
- *Plot Graphs* and the
- *Function Solver*

26.3.2 DATA EXPLORER

(Please remember that mouse click/drag on the visual area rotates/resizes the geometry)

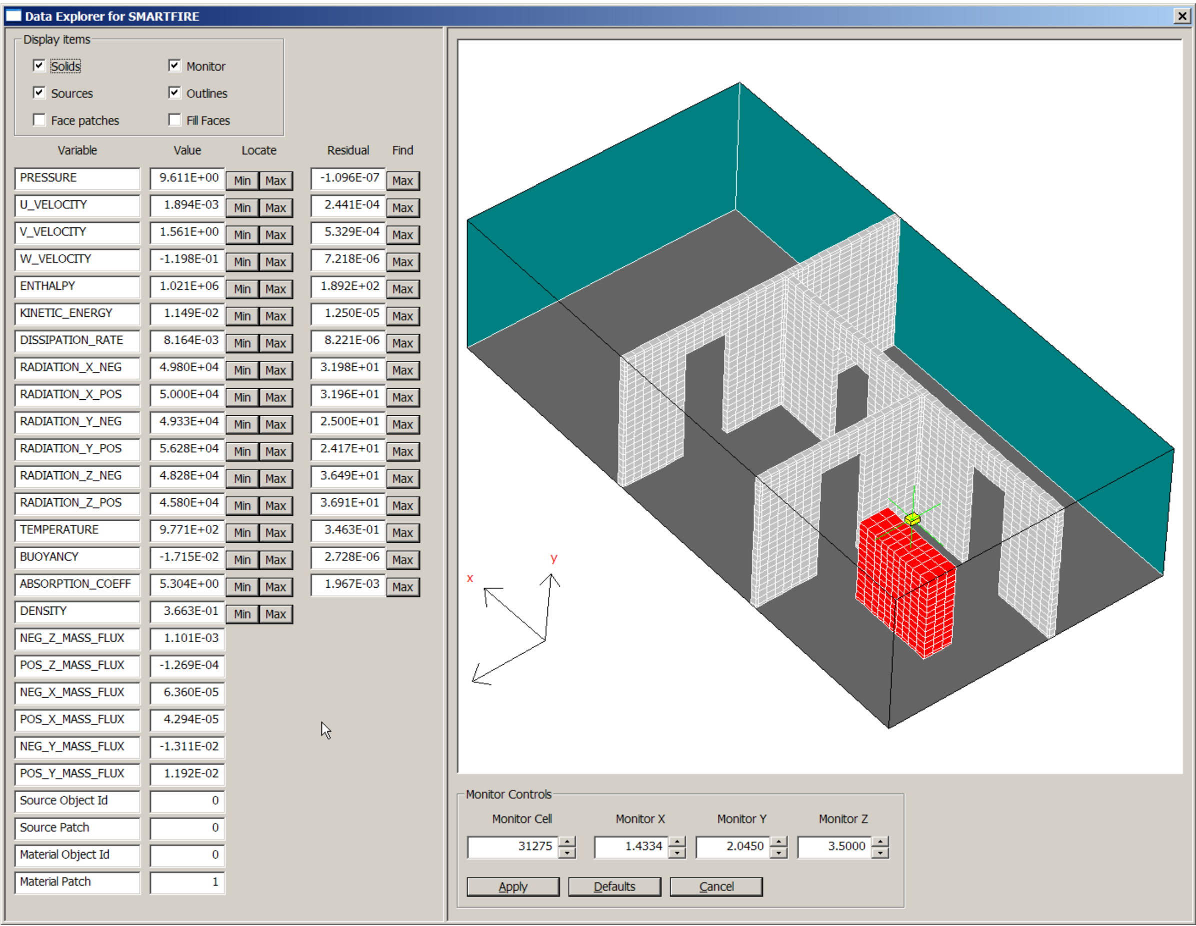

Figure 26-2 : The SMARTFIRE Data Explorer window.

Press the [Explore] button or select Display => Explore Data.

Expand the size of the explorer window as much as possible and drag the centre bar to the right in order to see all of the available data tools.

Use the Min- and Max- value and the Max- residual buttons to “find” the cell which has the maximum pressure, the minimum density, the maximum enthalpy residual and the maximum temperature. Record the cell number and its co-ordinates position (this allows comparison with a different mesh).

26.3.3 PLOT GRAPHS

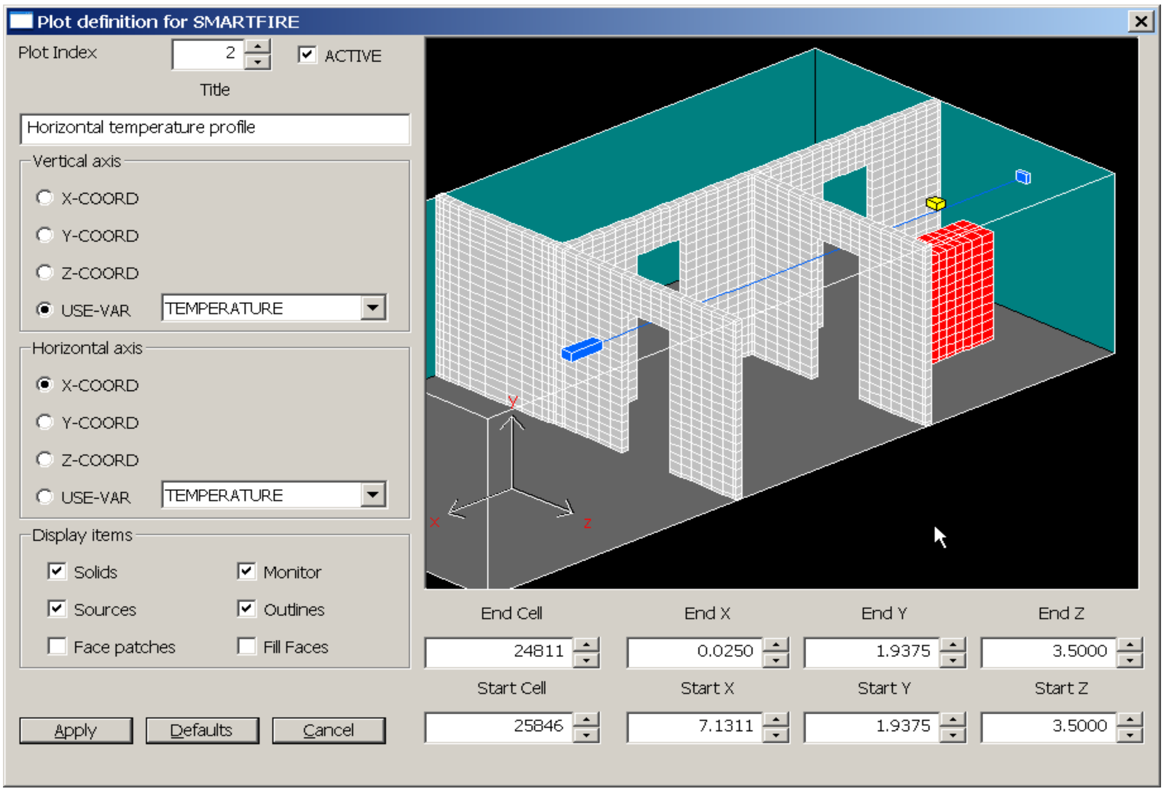

Figure 26-3 : The SMARTFIRE Plot Definition window.

Press the [Plots] button or select Display => Define Graphs.

Increment the Plot Index until you find a graph that is not yet active. *(This avoids changing existing plots from the Case Specification Environment).*

Activate this new graph definition using the [Active] tick box.

Name the plot as “horizontal T profile” so that you will recognise it.

Use the start and end coordinate positions to create a graph line that passes through the two rooms in the middle of the doors and just below the top of the doorway. i.e. between the points (0.0, 1.9, 3.5) and (7.0, 1.9, 3.5).

Select the vertical graph axis to be a [USE VAR] TEMPERATURE.

Select the horizontal graph axis to be [X-COORD].

[Apply] the settings and open the new graph plot window.

In order to save the graph plot data to file select [Results], tick the [Save Graph Data] check box and the press [Save Now]. This will save a file for each plot, and name them as:
"g#plot_index_#save_index.gpd".

26.3.4 FUNCTION SOLVER

Open the Function Solver window using:
Display => Function Solver.

Select and drag on the visual area to rotate the geometry so that you can easily see into the fire compartment.

Select region by volume and set the start and end coordinates so that the bright green outline box is just inside of the fire compartment walls.

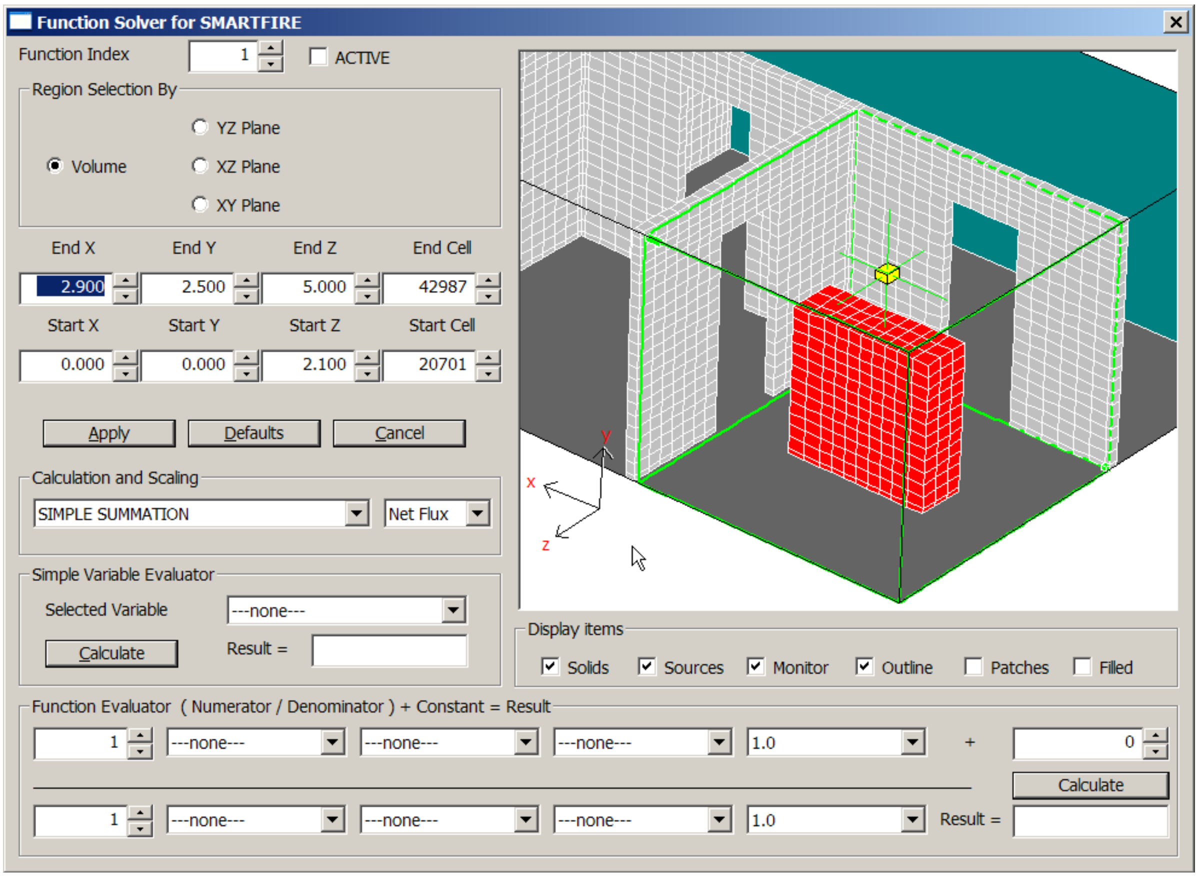

Figure 26-4 : The SMARTFIRE Function Solver window.

Set the calculation and scaling to VOLUMISED AVERAGE from the pull down menu.

Use the Simple Variable Evaluator area to choose the selected variable TEMPERATURE.

Press the [Calculate] button to see what the volumised average temperature is in the fire compartment. The result will be shown in the "Result =" box.

Also determine the average upper layer temperature. To do this you will have to decide on the average height of the hot layer. This is not always easy to do although you might find that cut-plane visualisations or plot graphs help to determine where there is a sharp change in temperature between the lower (cooler) layer and the upper (hotter) layer.

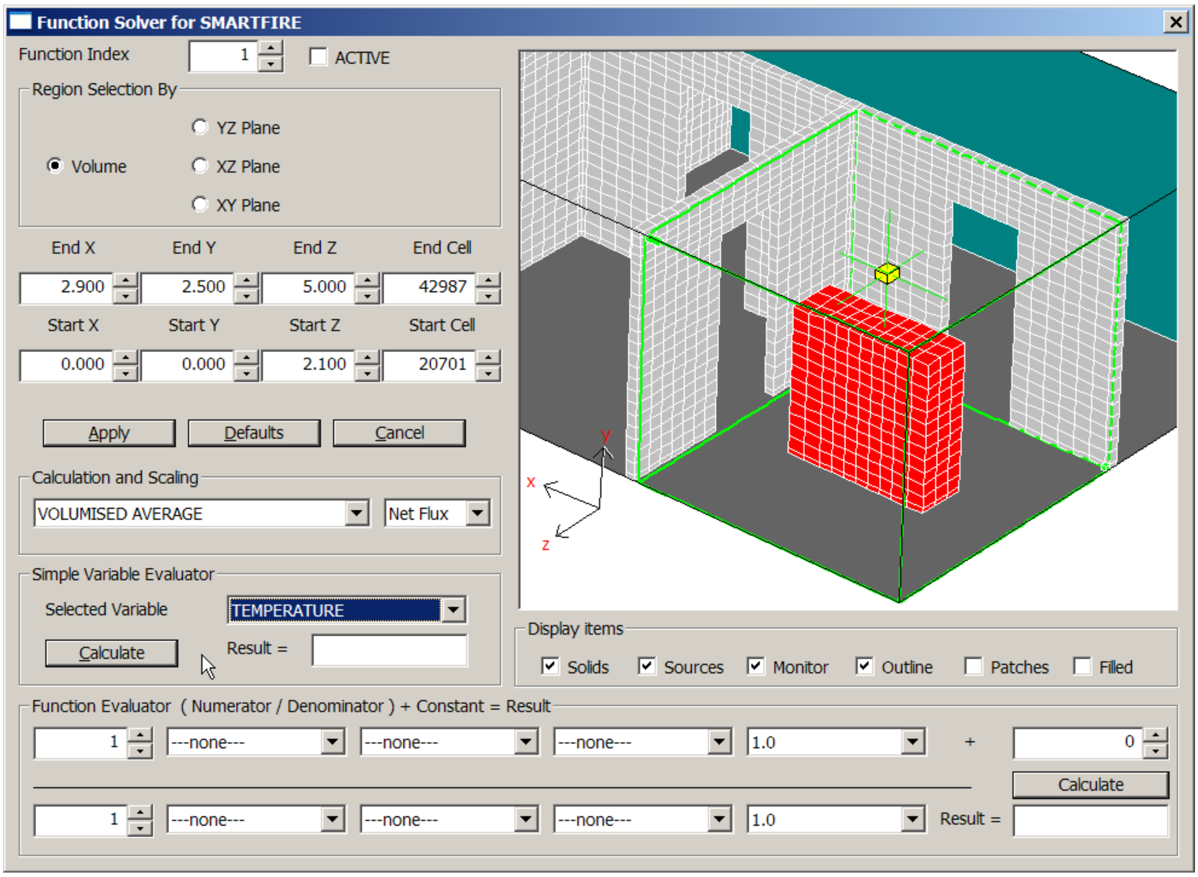

Figure 26-5 : Using the Function Solver window to select a sub-region.

Select and drag on the visual area to rotate the geometry so that you can easily see the X facing door into the fire compartment.

Select region by YZ Plane and set the plane x-coordinate and start- and end-coordinates so that the bright green outline box is just in the X-doorway of the fire compartment.

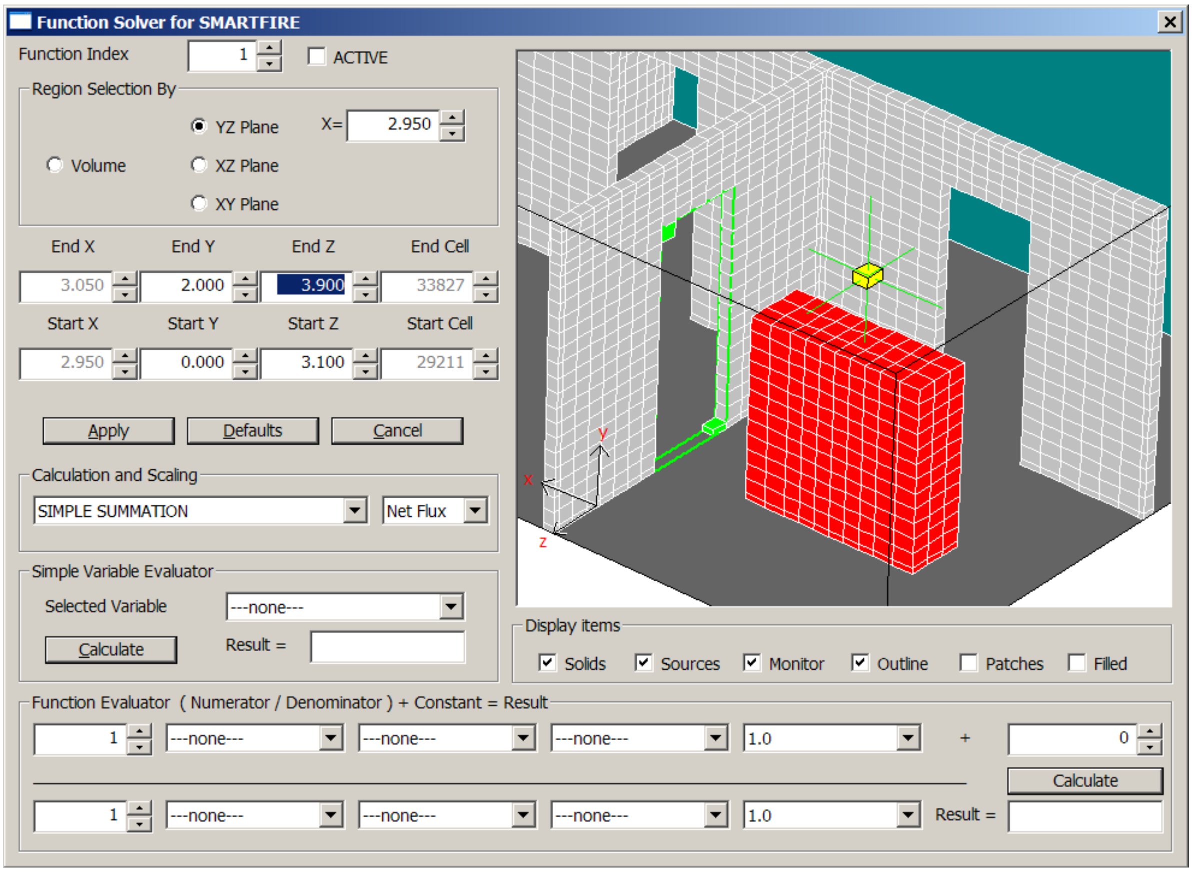

Figure 26-6 : Using the Function Solver to select the cells in a doorway.

Set the calculation and scaling to SIMPLE SUMMATION from the pull down menu.

Use the Simple Variable Evaluator area to choose the selected variable NET MASS FLUX.

Press the [Calculate] button to calculate the Net Mass Flux into the fire compartment through the X doorway. The result will be shown in the “Result =” box.

Now determine the mass inflow and mass outflow for the same door. Choose inflow or outflow from the variable list.

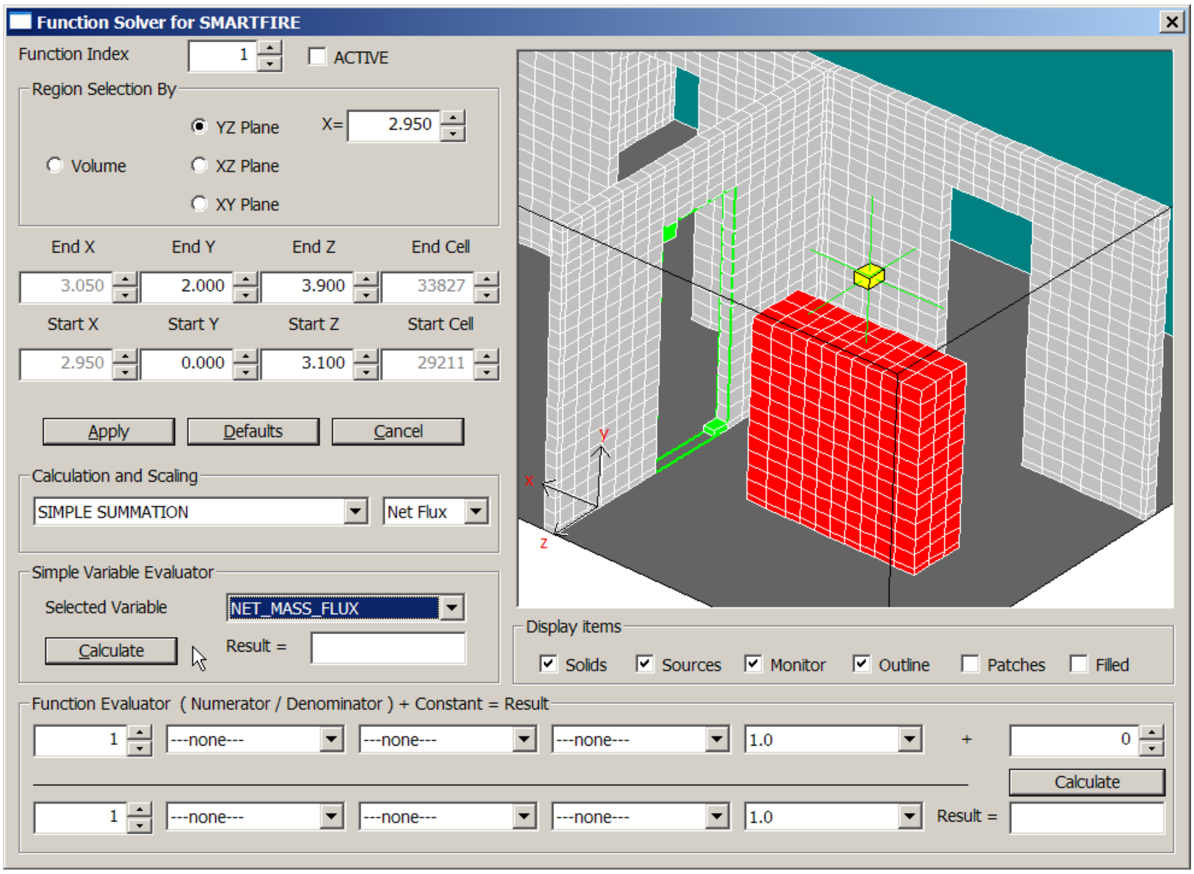

Figure 26-7 : Using the Function Solver to calculate the net flux through a doorway.

26.4 *TUTORIAL A3*

26.4.1 POST PROCESSING VISUALIZATION USING DATAVIEW

In this Tutorial you will need the ".VTK" and ".WRL" files that were saved during the Tutorial A1 simulation. If you have not saved any files you will need to repeat the processing of Tutorial A1 with the indicated result saving activated.

Start the ***SMARTFIRE*** DataView program by launching its icon from the desktop or from the link in the ***SMARTFIRE*** Case Specification Environment.

26.4.2 LOAD THE GEOMETRY

Select FILE => Import VRML and navigate to the Tutorial A1 folder which will be inside smartfire\work.

Load a ".WRL" file that was saved during the Tutorial A1 simulation.

This will show a semi-transparent display of the geometry of the scenario.

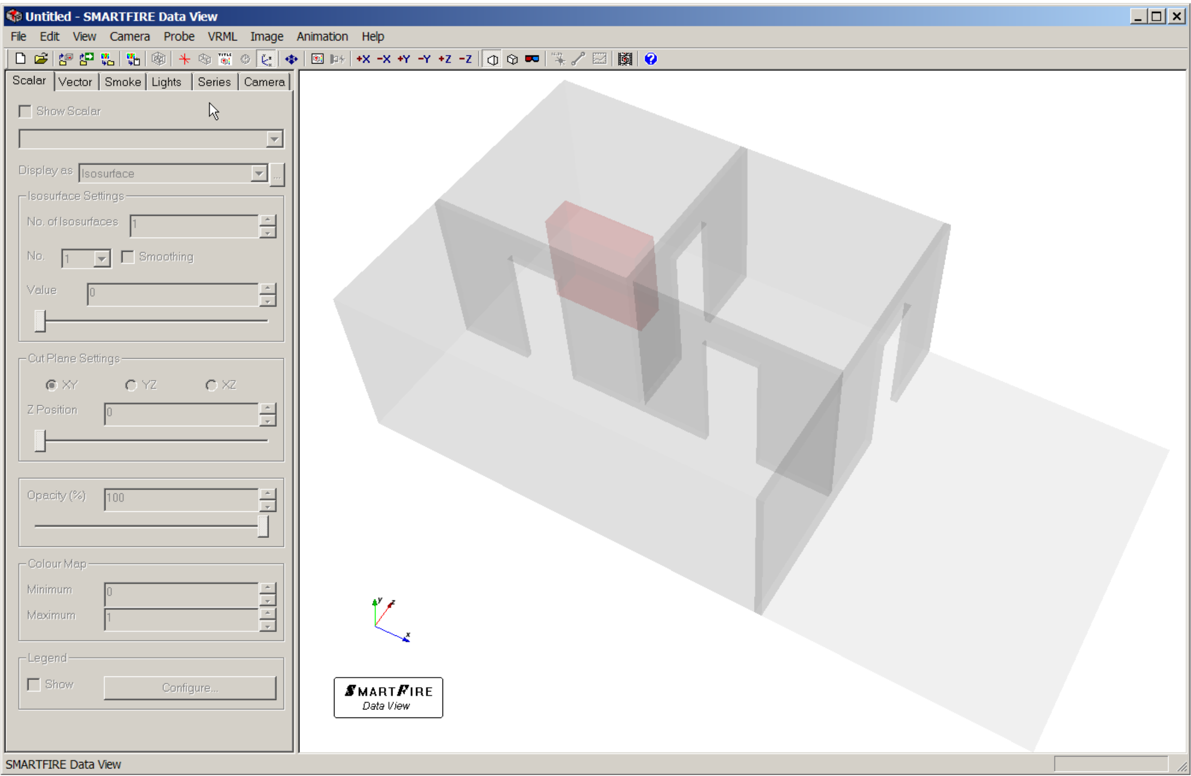

Figure 26-8 : The SMARTFIRE DataView program at start up.

Use mouse selection and dragging on the display area to rotate/resize/move the geometry to give the most appropriate (clearest) view

26.4.3 PREPARE FOR ANIMATION

Select the [Animation] panel and [Add Files…].

Select the files that you want to animate.

Highlight the last data file and select [Load File] to actually load the last data set.

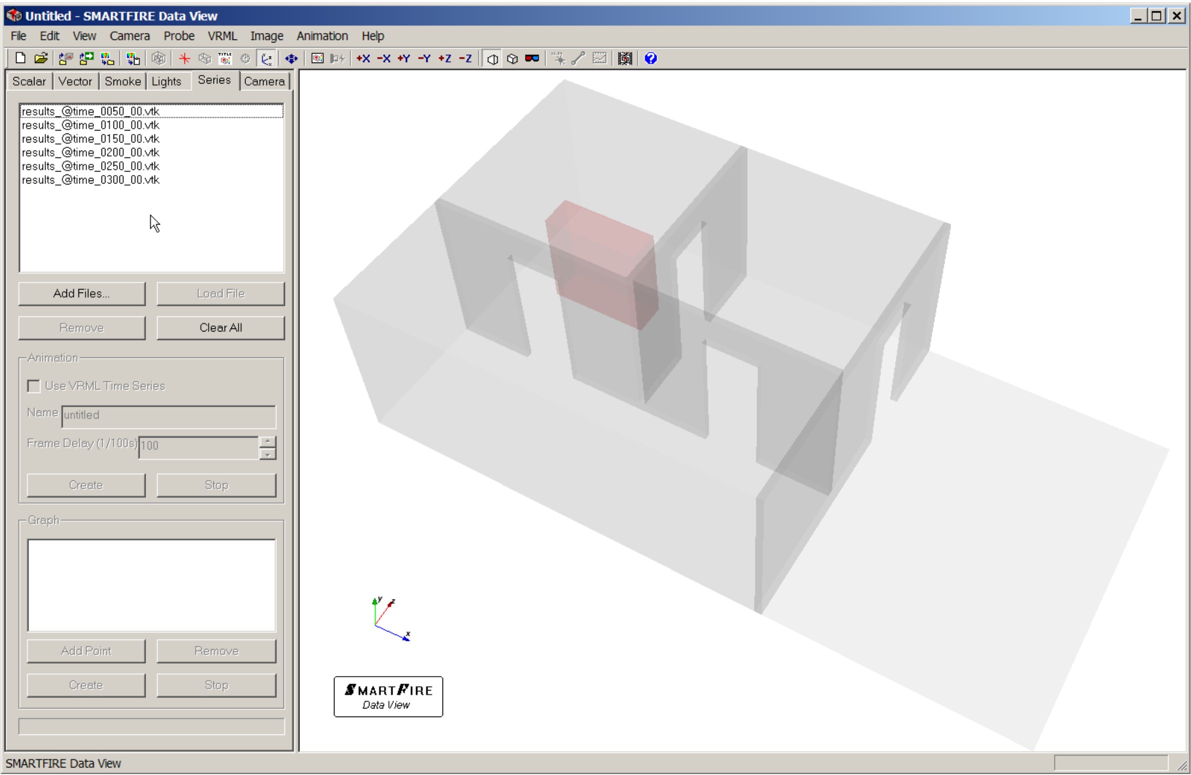

Figure 26-9 : DataView showing loaded geometry and animation list of data sets.

26.4.4 CREATE AN ANIMATED SCALAR VISUALIZATION

Select the [Scalar] panel and activate scalar iso-surface display and choose TEMPERATURE.

Choose an [iso-surface] and set a [value] for the surface (e.g. 700 K) and make the iso-surface semi transparent by setting the [Opacity] to 40 %. Add another iso-surface by changing the [No. of iso-surfaces] to 2 and setting the second iso-surface [value] to 900 K.

Create a title View => Title (e.g. “Tutorial A3”).

Return to the animation panel and enter a [name] (e.g. iso_anim) then press the [Create] button. The DataView program will now start to load each of the result (.vtk) files in turn and save a static image for each data set.

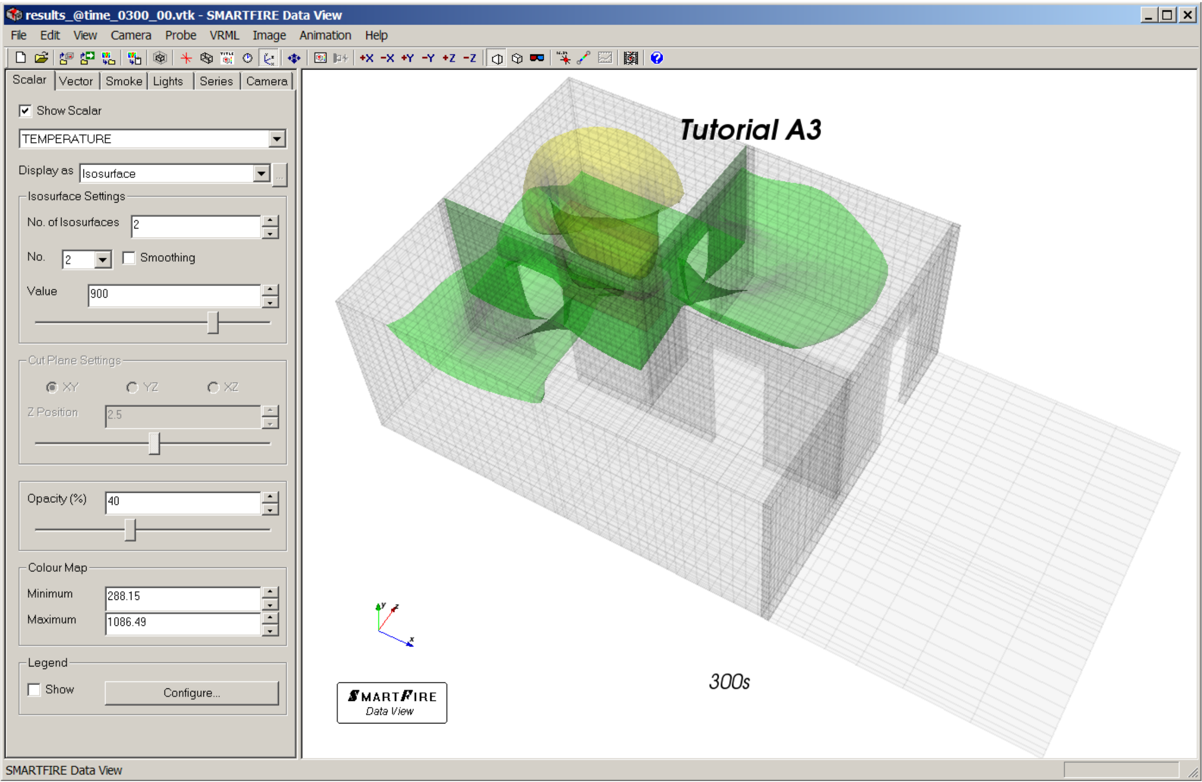

Figure 26-10 : DataView showing Temperature Iso-surface, title and time.

26.4.5 CREATE AN ANIMATED VECTOR VISUALIZATION

Once the previous animation has been successfully created, deactivate the iso-surface display and instead create an animation of a vector slice from the [Vector] panel.

Remember to change the title View => Title, if the previous title is now inappropriate for the new vector visualisation.

Once you have the required static vector display return to the animation panel and enter an animation [name] (e.g. vec_anim), then press the [Create] button.

26.4.6 VISUALIZATION NOTES

When investigating the configuration panels try setting/ticking the different options and note the effect on the visualization display.

You WILL NOT be able to visualize smoke for this tutorial because no smoke was saved in Tutorial A1.

You will be able to see smoke visualization from the data saved later in Tutorial A4.

Of course you can also simply use the DataView program to load and display a single data file, in which case use the File => Open option to select the required .VTK results file.

26.5 *TUTORIAL A4*

26.5.1 COMBUSTION MODELLING

In this Tutorial, the scenario from Tutorial A1 is extended to use triggered events (secondary fire ignition and opening of doors) and combustion modelling.

The geometry is as shown below:

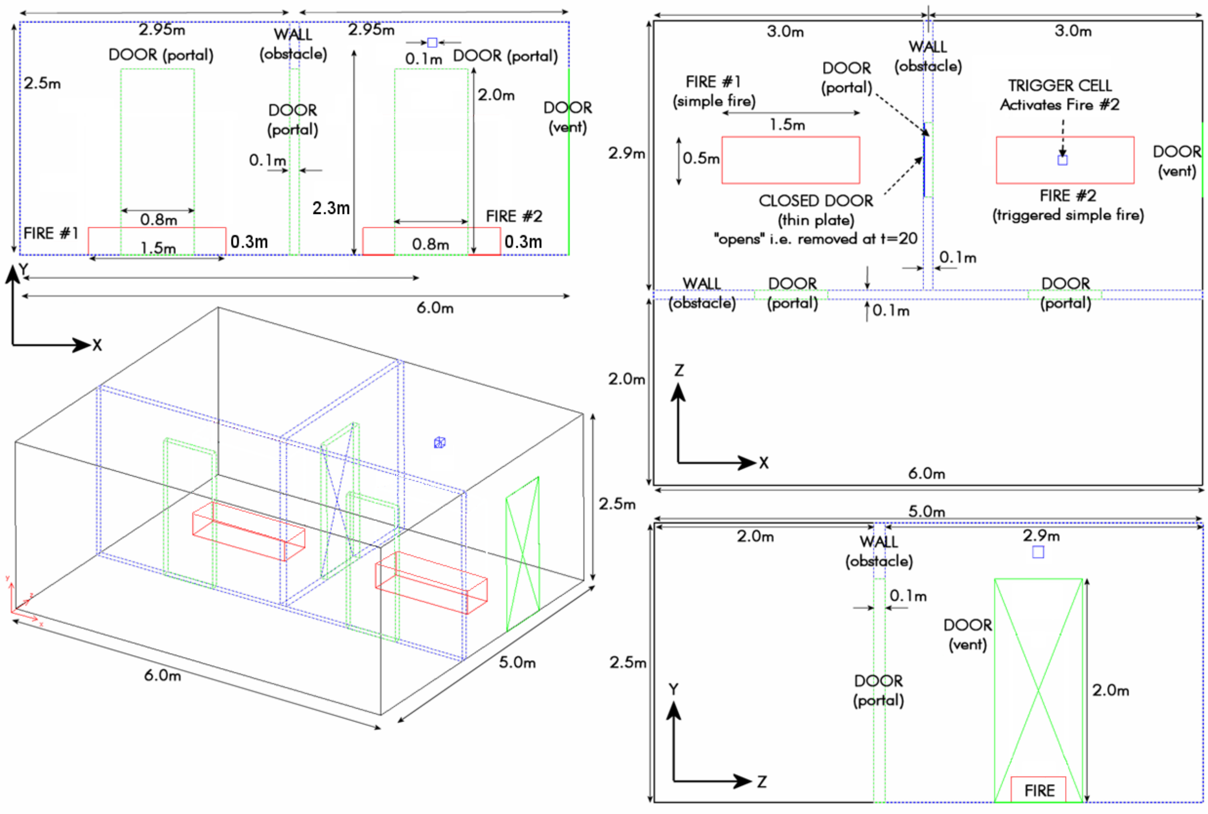

Figure 26-11 : Tutorial A4 Geometry showing modified fires, door and trigger cell.

26.5.2 MODIFY TUTORIAL A1 SCENARIO

<u>Load the base geometry and configure the new physics:</u>

Case is similar to Tutorial A1 with modified/additional objects, so load Tutorial A1 and save as "*tut_case_A4*" – to avoid any confusion.

Configure the Scenario => Problem Type with the following changes to the Tutorial A1 set-up (in the Problem Type configuration window):

Enable (a) Combustion model and (b) Smoke model

Leave all the default settings for the configuration of the Combustion-, Smoke- and Six Flux Radiation- models.

In order to use triggering, check that expert options are enabled.
Expert => Expert Options => Enable Expert Problem Set-up.

26.5.3 CASE SPECIFICATION

Create and modify the following objects:

Closed door [THIN PLATE in X direction]
at (2.95, 0.0, 3.1) size (0.0, 2.0, 0.8).

Trigger cell [TRIGGER CELL]
at (4.45, 2.3, 3.45) size (0.1, 0.1, 0.1).

Configure THIN PLATE "closed door" timed Activation as:
"Active until end" time of 20.0 seconds.
- Note that plate removal actually "opens" the doorway.

Configure the TRIGGER CELL for activation as:
TEMPERATURE GREATER THAN 500
- Note that all Temperatures are in Kelvin (i.e. K = oC + 273.15).

Configure the fuel release of the fire as follows:

Resize the [SIMPLE FIRE] so that the fire height is now 0.3m.

Set the current [SIMPLE FIRE] duration from 0.0 to 100.0 seconds.

Set simple t2 fuel curve
with C = 2.5e-6 (equivalent to peak 1 MW HRR fire).

Check the fire statistics to ensure that the fuel curve is correct.

Create and configure the Secondary Fire as:

Create a Secondary fire [SIMPLE FIRE]
at (3.75, 0.0, 3.25) size (1.5, 0.3, 0.5).

HINT: Easy way to create this fire is to "clone" the first fire and change position. The fuel (or HRR) curve will be the same as for the copied fire source.

Check that the fuel mass release is the same as for the first fire.

Configure fire activation as "Activated by Trigger" and select the only trigger name from the pull-down list of available trigger objects.

Configure regular data captures as follows:

Check that results will be saved by times rather than time step numbers by ticking:

Expert => Expert Options => Use times for data saves

Select result saves for every 10 seconds of simulated time using:

Scenario => Output Control => Use transient result saves, with the save interval set to 10

26.5.4 MESH AND RUN THE SIMULATION

Configure graph plot creation (data monitoring)

as for Tutorial A1.

Meshing options:

Check case saved as "*tut_case_A4*" and select Run => Create Mesh.

Choose to ignore overlaps since they are all intentional.

Select the geometry type as "Standard Room Geometry".

Stop the meshing system from adding an extra extended region in Y+ direction.

Choose a Cell Budget for Meshing as "Recommended mesh" with the "Refine" option enabled.

Accept the mesh (this creates script files for the SMARTFIRE CFD Engine).

Run the simulation:

Select the Run => Run CFD Engine option.

Wait for the SMARTFIRE CFD Engine to initialise and open fully.

Press the [GO/Run] button to start the simulation.

Stop and examine data or settings at any time with the [Stop] button.

26.5.5 RESULTS ANALYSIS

Results/Observations:

Examine the "*tut_case_A4.log*" file to see when the second fire ignited.

HINT search for "CHANGED VOLUME PATCH" and examine the sweep information just above it to see when the event actually triggered.

What is the maximum temperature at the end of the 3 minute simulation?

Take note of the level of convergence that was achieved.

Do you think that this is an acceptable level of convergence?

Did you notice how the convergence affected when the door was opened and when the second fire ignited? You should notice that the simulation becomes less stable when the door first opens.

As with Case A1, Note the temperature and horizontal velocity distribution through the vertical centre line of each of the doors and the vertical temperature distribution in the centre of the corridor. This data should be available for 30 seconds, 1, 2 and 3 minutes into the simulation.

Compare these results with the results from the previous simulation. With combustion and a radiation model enabled, you should initially see lower temperatures than when using a heat release rate model. The temperatures will rise significantly when the second fire has ignited.

26.6 *TUTORIAL A5*

26.6.1 CAD IMPORT TUTORIAL

This tutorial introduces the ***SMARTFIRE*** Scenario Designer and uses it to generate a scenario from part of a large scale building plan that is a DXF 2D floor plan layered CAD drawing.

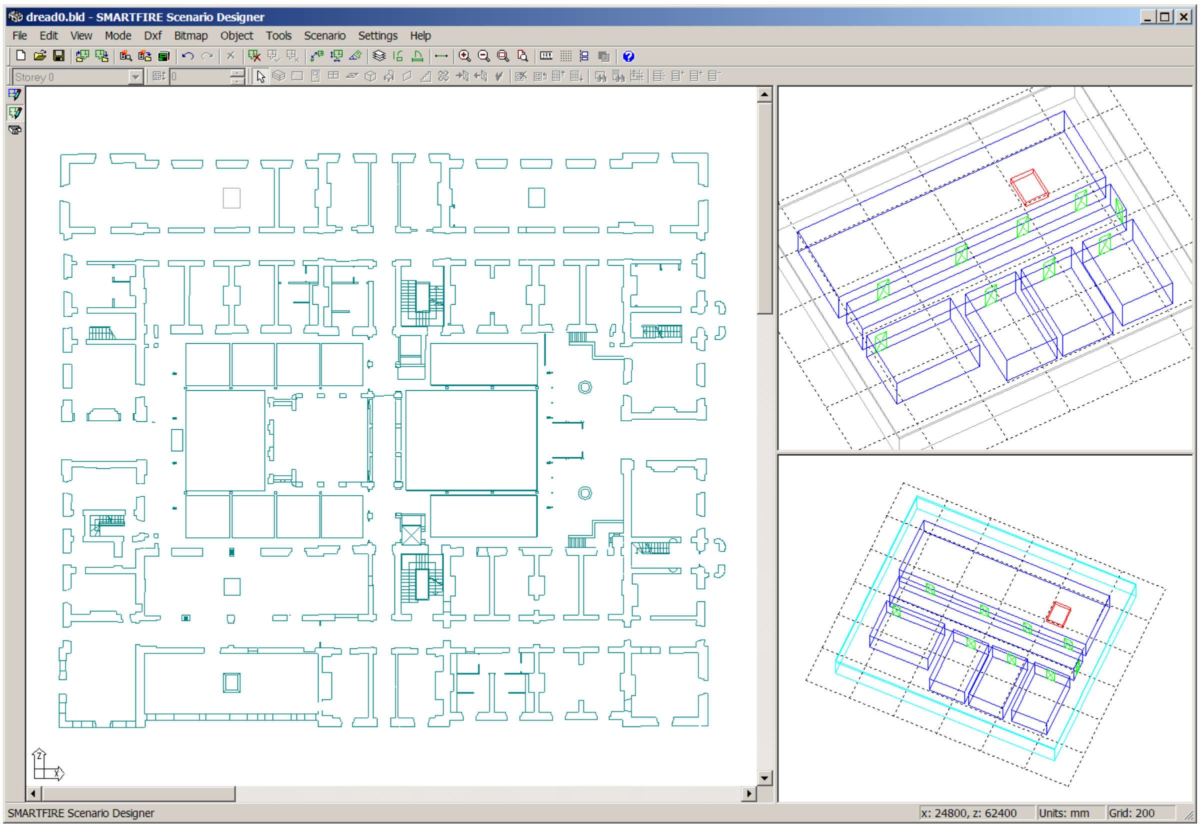

Figure 26-12 : SMARTFIRE Scenario Designer showing loaded CAD floor plan.

<u>Load the supplied CAD floor plan into the Scenario Designer:</u>

Run the Scenario Designer and select File => Import DXF…

Load the file "\smartfire\tutorial\CAD_DXF_geometry\dread0.dxf".

Ensure Settings => Units are millimetres.

Set the Settings => Grid Spacing to 500.

Use DXF => Layers to Disable *NEW_WALLS* and *STAIRS.*

26.6.2 CREATE FLOOR PLAN

Create a Scenario Designer Floor Plan:

Select the Object => Storey and draw a selection box around a part of floor plan from the top left corner to about 1/3 across and 1/3 down, which encloses the top-left wide room, part of the corridor and the four rooms below it.

Go into Edit DXF Floor Plan mode and add a line across the corridor to help the room locator find the “open ended” corridor.

Disable the box in the top-left long room by selecting it and then
DXF => Disable Selected (since this would confuse the room locator).

Return to Edit Building Model Floor Plan mode.

Find / Add rooms to the Scenario:

Check the Object => Default Properties.

Check “storey” height=2.5m, “door” height=2.0m and width=0.8m.

Select Tools => Locate Rooms and draw a selection box around the whole storey.

Choose OK to find rooms and draw a line to indicate the “width” of a reasonably wide doorway.

Room locator should find 6 rooms as in the floor plan. Add these to the scenario. You may need to edit the corridor “room” object so that it is not touching the top room object by dragging the edge of the room slightly.

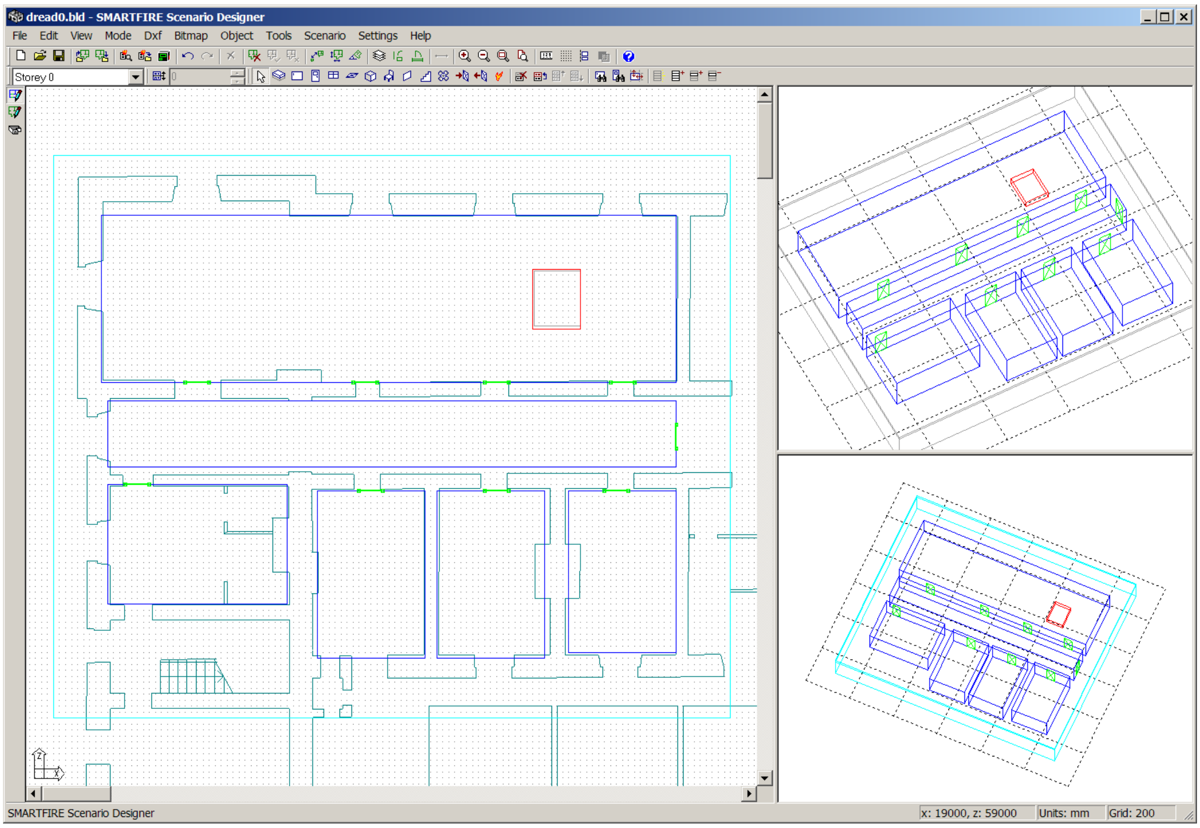

Figure 26-13 : Scenario Designer showing zoomed sub-region for scenario model.

<u>Add other objects to the Scenario:</u>

Select Object => Fire and draw a “box” for the fire in the long room.

In selection mode (arrow) right click on Fire, select Properties and set height to 1.5m in the properties pop-up menu.

Select Object => Door and click on the plan to add doors to the gaps in the walls.
Hint: select middle of the gap and only needed on one edge of a wall.

Use Scenario => Add Building to add all objects to the scenario.

Select File => Create Smartfire Simulation.

Give the case a name (e.g. “tutorial_a5” or “dread0_t5”) and proceed to the Case Specification Tool where the scenario can be finalized and meshed.

Also you might like to try to create a scenario from another section of the floor plan.

26.7 *TUTORIAL A6*

26.7.1 USING PARALLEL SMARTFIRE

The following tutorial is intended to take you through building, running and analysing a CFD based fire simulation using parallel SMARTFIRE.

The case involves a small two room geometry connected by a corridor. One of the rooms is vented to the outside. CAD import will be used to construct the case from a floor plan diagram.

The tutorial uses the Scenario Designer to build the geometry, the Case Specification Environment to configure and mesh the case and the parallel version of the CFD Engine to run the scenario in a variety of configurations.

For comparison purposes, it is essential that you ensure that all the settings for the simulations are identical. (The same as for Tutorial A1).

You need to ensure that both MPICH and SMARTFIRE are installed correctly on all the machines that will be used in the parallel computation.

26.7.2 LOAD DXF FLOOR PLAN

<u>Load a DXF geometry floor plan into the Scenario Designer:</u>

This part of the tutorial is intended to practice the use of the CAD interface (i.e. the ***SMARTFIRE*** Scenario Designer) to generate the scenario to be used in parallel ***SMARTFIRE***.

Run the Scenario Designer and select File => Import DXF…
and choose to load the file:
“\smartfire\tutorial\CAD_DXF_geometry\cad_geometry.dxf”

The floor plan should appear as in the figure below:

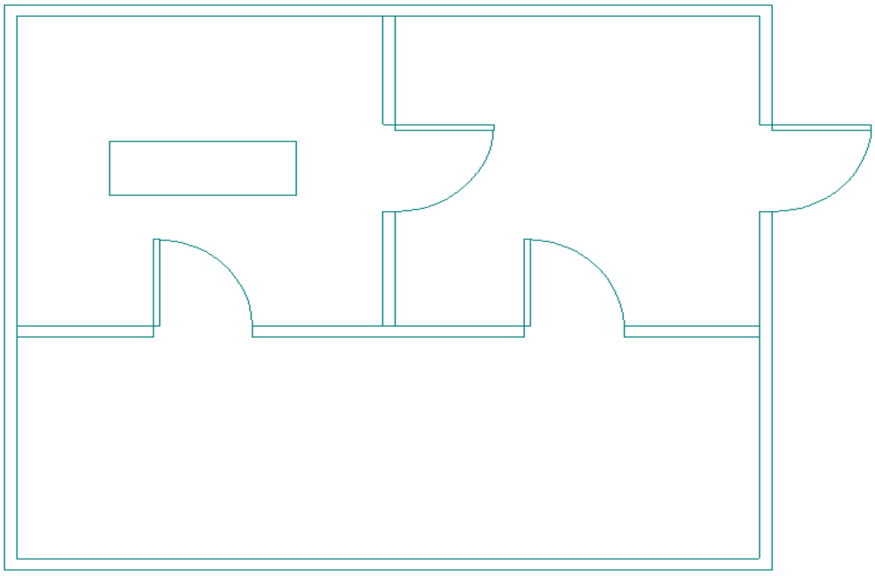

Figure 26-14 : Simple CAD floor plan representing Tutorial A1 Scenario.

26.7.3 CREATE SCENARIO

<u>Create the scenario in the Scenario Designer:</u>

Ensure Settings => Units are metres and set the Settings => Grid Spacing to 0.05.

Use DXF => Layers to delete the *DOORS* and disable the *FURNITURE.*

Use Object => Storey, Draw storey box around the whole floor plan.

Check the Object => Default Properties.

Check “storey” height=2.5m, “door” height=2.0m and width=0.8m.

Select Locate => Rooms, Draw a search region around the geometry.

Choose OK to find rooms, Draw a line to indicate doorway “width”.

Tool should find 3 rooms as in the floor plan.

Select Object => Fire. Draw a box over the greyed furniture item.

In selection mode (arrow) right click on Fire, select Properties and set height 1.5m.

Select Object => Door, click to add doors to the gaps in the walls.
Hint: select the middle of a gap and only on one edge of a wall.

You should arrive at the following floor plan:

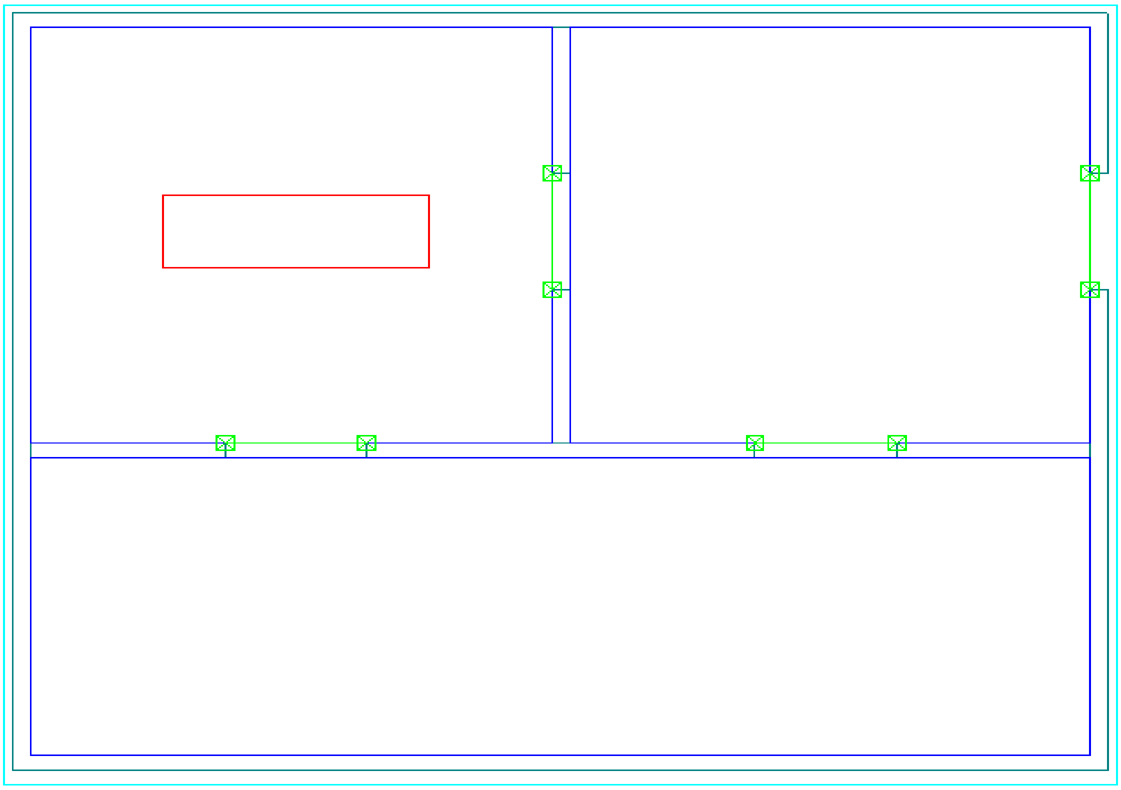

Figure 26-15 : Scenario Designer floor plan with door and fire objects.

Create the SMARTFIRE Scenario:

Select Scenario => Add storey to add this storey to the model.

Select File => Preview SMF – this should look almost identical to Tutorial A1.

Select File => Create SMARTFIRE simulation to pass the scenario to SMARTFIRE

Name the case "*tut_case_A6*" and this will save the "*tut_case_A6.smf*" file in a new "smartfire\work*tut_case_A6*" folder.

The *Case Specification Environment* will launch automatically with the geometry loaded.

26.7.4 RUN THE PARALLEL CFD SIMULATION

Run the simulation in the parallel MPI version of Smartfire:

Run parallel SMARTFIRE using:

Run => Run Parallel CFD Engine.

This will launch the parallel launcher for the configuration of the parallel run.

Ensure that the data is saved at the correct time intervals (i.e. every 30s) for comparison with data from Tutorial A1.

26.7.5 EXERCISES FOR PARALLEL SMARTFIRE TUTORIAL

<u>Results/Observations:</u>

What is the maximum temperature at the end of the 3 minute simulation?

Take note of the level of convergence that has been achieved.

Have you reached an acceptable level of convergence?

How does the level of convergence compare with Case A1?

Are there differences in the convergence quality of the serial and parallel results?

Ensure that you have saved the temperature and horizontal velocity distribution through the vertical centre line of each of the doors and the vertical temperature distribution in the centre of the corridor at 30 sec, 1, 2 and 3 minutes into the simulation.

How do these results compare for the serial and parallel cases?

How do the results compare for the coarse and fine mesh? Also compare cell budgets.

Are the coarse results acceptable?

What conditions would you place on this level of acceptability?