Tutorial: Using Solar Load Model for Indoor Ventilation

Introduction

The purpose of this tutorial is to provide guidelines and recommendations for setting up and solving an indoor ventilation problem of a building using solar load model. The problem is initially solved excluding the radiation effects, and then radiation model is included in the calculation to study the effect of radiant heat exchange between the internal surfaces.

Pre-requisites

This tutorial assumes that you are familiar with the FLUENT interface and that you have good understanding of basic setup and solution procedures. In this tutorial, you will use solar load model and radiation model, so you should have some experience with them. This tutorial does not cover the mechanics of using these models, but focuses on the applications of this model to solve the indoor ventilation problem.

Problem Description

The problem considered is ventilation of the reception area of Fluent Europe's office at Sheffield, England (Figure 1). The front wall of the reception is almost fully glazed from the ground floor to the ceiling of the first floor level. There is also a small glazed area on the roof above the second floor landing.

This tutorial looks at the typical expected loads in the middle of summer on a normal day. The adjoining rooms and offices are air-conditioned and maintained at a constant temperature of around 20 ^{0}C. So heat will be transferred through the internal walls to these rooms. Some heat will also be transferred to the floor. As the floor is a concrete base it is assumed to have substantial thermal mass and a fixed temperature. Outside conditions are considered to be pleasant and normal at 25 ^{0}C. An external heat transfer coefficient of 4 W/m^2K is used to capture convective heat transfer from outside the building. There is an air cooling unit located behind the receptionist's desk.

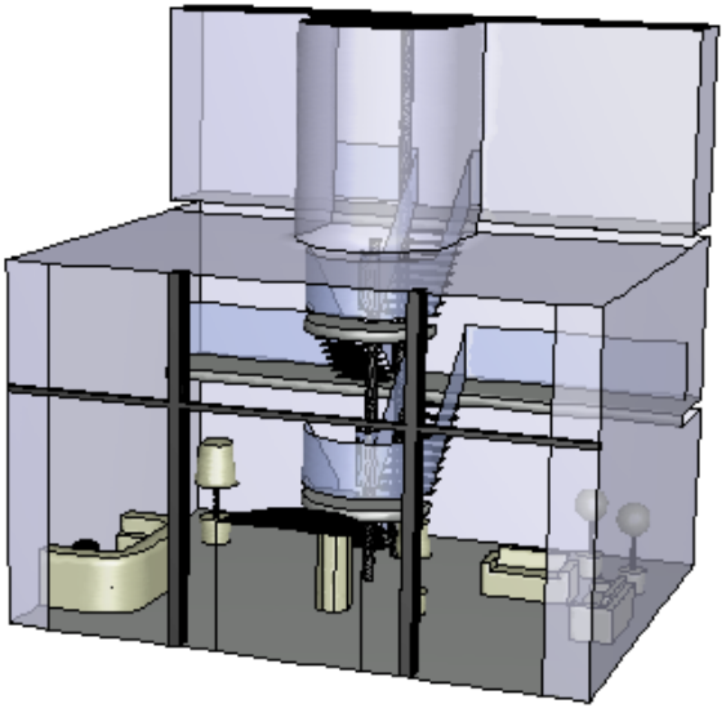


Figure 1: Problem Schematic—Reception Area

Preparation

1. Copy the files fel_atrium.msh to your working folder.
2. Start the 3D (3d) version of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the mesh file fel_atrium.msh.

   File ⟶ Read ⟶Case...

2. Check the grid.

   Grid ⟶Check

3. Display the grid (Figure 2).

   Display ⟶Grid...

(a) Select Feature in the Edge Type group-box.

(b) Click Display and close the Grid Display panel.

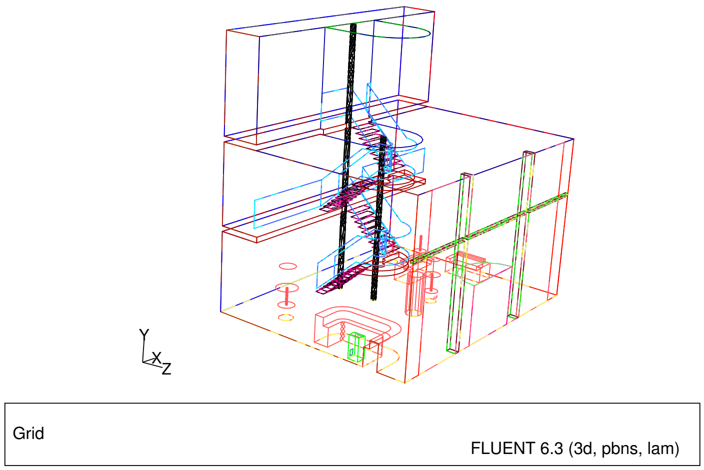


Figure 2: Graphics Display of the Grid (Features)

We will follow a naming convention for labeling the grid. All the walls are prefixed with 'w' and all velocity inlets are prefixed with 'v'.

Step 2: Units

1. Change the default unit for temperature to centigrade.

   Define ⟶Units...

   (a) Select temperature from the Quantities list.

   (b) Select c from the Units list and close the Set Units panel.

Step 3: Models

1. Define the solver settings.

   Define ⟶ Models ⟶Solver...

   (a) Select Pressure Based in the Solver group-box.

   (b) Select Green-Gauss Node Based in the Gradient Option group-box.

   This provides improved resolution of gradients, particularly with coarse or skewed cells.

   (c) Click OK to close the Solver panel.

2. Enable the effects of gravity.

   This is because the bulk of the flow is driven by natural convection.

   Define ⟶Operating Conditions...

   (a) Enable Gravity.

   (b) Enter -9.81 for Y in the Gravitational Acceleration group-box.

   (c) Click OK to close the Operating Conditions panel.

3. Enable k-epsilon turbulence model.

   Define ⟶ Models ⟶Viscous...

   The flow is expected to be turbulent, so an appropriate turbulence model is required.

   (a) Enable k-epsilon (2 eqn) from the Model list.

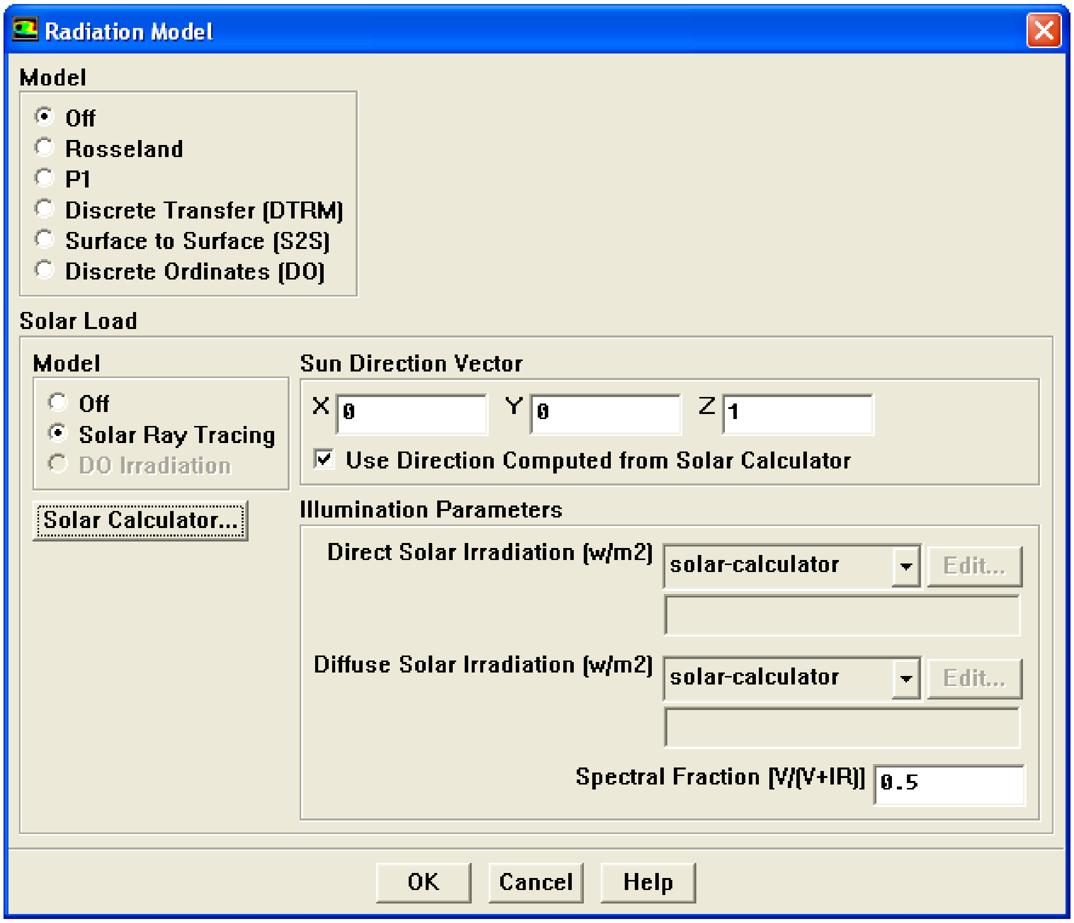
   (b) Enable RNG in the k-epsilon Model group-box.

   (c) Enable Full Buoyancy Effects in the Options list.

   (d) Click OK to close the Viscous Model panel.

4. Enable the solar load model.

   Define ⟶ Models ⟶ Radiation...



   (a) Enable Solar Ray Tracing in the Solar Load group-box.

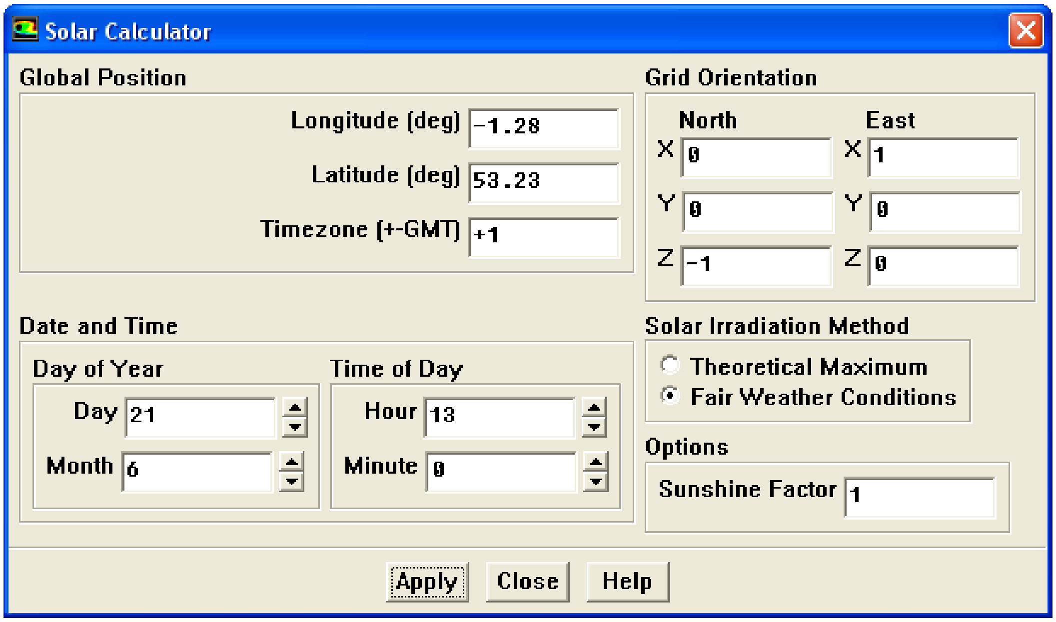
   (b) Retain the default selection of Use Direction Computed from Solar Calculator under Sun Direction Vector.

   (c) Select solar-calculator from the Direct Solar Irradiation and Diffuse Solar Irradiation drop-down lists.

   (d) Retain the default value of `0.5` for Spectral Fraction [V/(V+IR)].

   A value of 0.5 specifies 50% split between long (IR) and short (V) wave radiation.

   (e) Click the Solar Calculator... button in the Radiation Model panel.



i. Enter `-1.28` deg for Longitude and `53.23` deg for Latitude in the Global Position group-box.

ii. Enter `+1` for Timezone in the Global Position group-box.

A value of 1 specifies British summer time.

iii. Enter `0, 0, -1` for North and `1, 0, 0` for East in the Grid Orientation group-box.

iv. Retain default values for Date and Time.

Default values are set to peak summer conditions.

v. Retain the default selection of Fair Weather Conditions in the Solar Irradiation Method group-box.

These conditions are representative of a fine day with little cloud cover.

vi. Retain the default value of `1` for the Sunshine Factor.

You can assume that there is no additional cloud cover.

vii. Click Apply and close the Solar Calculator panel.

Information will be printed in the FLUENT *console summarizing the diffuse and direct components that will be used to calculate the load.*

(f) Modify other model parameters that are available only through the TUI.

i. Retain default value for ground-reflectivity.

```
/define/models/radiation/solar-parameters> ground-reflectivity

Ground Reflectivity [0.2]
```

When the solar calculator option is selected for diffuse radiation, the radiation reflected from the ground will be added to the total diffuse background radiation. The amount of radiation reflected from the ground depends on its reflectivity which can be set through this parameter. The default value of 0.2 is reasonable.

ii. Reduce the scattering fraction to a value of `0.75`.

```
/define/models/radiation/solar-parameters> scattering fraction

Scattering Fraction [1] 0.75
```

The solar load ray trace model provides directional loading only to the first opaque surface that it reaches. It does not undertake further ray tracing to account for reradiation due to reflection or emission. It does not discard the reflected component of radiation, but distributes it across all participating surfaces.

The scattering fraction, i.e., the amount of the reflected component which is distributed, is set to a default value of 1. This means that all the reflected radiation will be distributed inside the domain. If the building has a large glazed surface area, a significant amount of the reflected component is expected to be lost through external glazing. In such cases, reduce scattering fraction accordingly.

iii. Activate sourcing of energy into an adjacent fluid cell.

```
/define/models/radiation/solar-parameters> sol-adjacent-fluidcells

sol-adjacent-fluidcells? [no] yes
```

The solar load model calculates energy sources for every face subjected to a solar load. By default, this energy will be sourced into an adjacent shell conduction cell, if available (if planar conduction is being used). Otherwise, it will be put into an adjacent solid cell.

If neither conduction cell nor solid cell is adjacent, it will be sourced into the adjacent fluid cell. However, if the mesh is too coarse to accurately resolve wall heat transfer (as is frequently the case with building studies), then it is preferable to put the heat straight into the fluid cell every time. This helps reduce the likelihood of predicting unnaturally high wall temperatures while still achieving the energy transfer into the room.

Step 4: Materials

Define ⟶Materials...

In this step you will modify fluid properties for air and solid properties for steel. You will also create new materials (glass and a general insulating building material) in the setup.

1. Modify the parameters for air.
   (a) Select boussinesq from the Density drop-down list and enter `1.18` as the value of density.

   A value of 1.18 kg/m^3 specifies the appropriate density of air at 25 ^0C and 1 atm.

   This setting is more stable for problems with natural convection, and is valid over relatively small variations in temperature. In general, if the temperature range is more than 10-20% of the absolute temperature (in Kelvin) then another approach should be considered.

   (b) Enter `0.00335` K^{-1} for the Thermal Expansion Coefficient and click Change/Create.

   Assuming an ideal gas relationship this is the inverse of the absolute temperature (in Kelvin). For air at 25 ^0C, it is 0.00335 K^{-1}.

2. Copy the material steel.
   (a) Click the Fluent Database... button to open the Fluent Database materials panel.
   (b) Select solid from the Material Type drop-down list and select steel from the Fluent Solid Materials drop-down list.

   Ensure that all other materials have been deselected from the list.

   (c) Click Copy and close the Fluent Database materials panel.

3. Create a material called glass.
   (a) Select steel from the Fluent Solid Materials drop-down list.

   Rename it as glass.

   (b) Enter `glass` as the material Name in the Materials panel.
   (c) Specify the following parameters:

Parameter	Value
Density (kg/m3)	2220
Cp (j/kg-k)	830
Thermal Conductivity (w/m-k)	`1.15`

   (d) Click Change/Create and click No in the question dialog box asking whether to overwrite existing materials.

4. Similarly, create another material called `building-insulation` with following properties:

Parameter	Value
Density (kg/m3)	10
Cp (j/kg-k)	830
Thermal Conductivity (w/m-k)	`0.1`

Step 5: Boundary Conditions

Define ⟶Boundary Conditions...

By default, all the internal and external boundaries participate in the solar ray tracing and are opaque. All the wall in this model participate in solar ray tracing.

1. Set the boundary conditions for solid-steel-frame.

   (a) Select Steel as the Material Name.

   The steel frame is actually a hollow section. In this case, it is represented as a solid section. The properties of steel have been retained only for simplicity.

   (b) Click OK to close the Solid panel.

2. Set the boundary conditions for w_floor.

   (a) Click the Thermal tab and enable Temperature in the Thermal Conditions group-box.

   (b) Click the Radiation tab and enter `0.81` for Direct Visible and `0.92` for Direct IR in the Absorptivity group-box.

   (c) Click OK to close the Wall panel.

3. Set the boundary conditions for external gazed wall (`w_south-glass`).

   (a) Click the Thermal tab and enable Mixed in the Thermal Conditions group-box.

   The thermal condition is set to Mixed *to account for external convective heat transfer and external radiant losses from the glazing unit.*

   (b) Specify following parameters:

Parameter	**Value**
Heat Transfer Coefficient (w/m2-k)	4
Free Stream Temperature (c)	22
External Emissivity	0.49
External Radiation Temperature (c)	-273

   The value of External Radiation Temperature *is not set to the outside background temperature value because the incoming radiation is already being supplied by the solar load model.*

   (c) Select glass from the Material Name drop-down list and enter `0.01` m for Wall Thickness.

   This is a double-glazed wall and should have material properties which account for the gas space between the two panes. But you can ignore this in the first instance.

   (d) Click the Radiation tab and select semi-transparent from the BC Type drop-down list.

      i. Enter `0.49` for Direct Visible, Direct IR and Diffuse Hemispherical in the Absorptivity group-box.

ii. Enter 0.3 for Direct Visible and Direct IR and 0.32 for Diffuse Hemispherical in the Transmissivity group-box.

(e) Click OK to close the Wall panel.

4. Similarly set the boundary conditions for w_doors-glass and w_roof-glass.

5. Set the boundary conditions for w_roof_solid.

   The roof is an opaque external surface that receives a solar load from the outside. As it is well insulated, we can assume that there is little heat transfer across it. The default thermal condition of zero heat flux can be used.

   (a) Retain the default settings for the Thermal tab.

   (b) Click the Radiation tab.

   Specify the values as per the radiant properties for walls mentioned in the table in Appendix.

   i. Ensure that BC Type is set to opaque and Participates in Solar Ray Tracing is enabled.

   ii. Enter 0.26 for guiDirect Visible and 0.9 for Direct IR in the Absorptivity group-box.

   (c) Click OK to close the Wall panel.

6. Set the boundary conditions for w_steel-frame-out.

   (a) Click the Thermal tab and enable Mixed in the Thermal Conditions group-box.

   You need to account for external solar loading and convection. The computed solar loads are not applied to the outside surface of any opaque surfaces located at the boundary of the domain. Instead, they need to be imposed using the thermal conditions.

   The direct solar load calculated by the solar calculator is 857.5 w/m^2 at a direction vector of (0.0275, 0.867, 0.496). The direct south-facing vertical surface loading comes out to be 425 w/m^2. Add to this the diffuse vertical surface loading of 134.5 w/m^2 and ground reflected radiation of 86 w/m^2 to get a total vertical surface loading of 645.5 w/m^2. This equates to an effective radiation temperature of 326.65 K or 53.5 ^0C.

   You can ignore the material name and wall thickness as the wall thickness is represented explicitly.

   i. Specify the values for the parameters as per the following table:

Parameter	Value
Heat Transfer Coefficient (w/m2-k)	4
Free Stream Temperature (c)	25
External Emissivity	0.91
External Radiation Temperature (c)	53.5

(b) Retain the default settings under Radiation tab.

This wall boundary will not face any incident radiation as is not adjacent to air.

(c) Click OK to close the Wall panel.

7. Set the boundary conditions for w_steel-frame-in.

(a) Retain the default settings in the Thermal tab.

(b) Click the Radiation tab.

Specify the values as per the radiant properties for dark grey gloss material mentioned in the table in Appendix.

i. Ensure that BC Type is set to opaque and Participates in Solar Ray Tracing is enabled.

ii. Enter 0.78 for Direct Visible and 0.91 for Direct IR in the Absorptivity group-box.

(c) Click OK to close the Wall panel.

8. Set the boundary conditions for w_north-wall.

(a) Click the Thermal tab and enable Convection in the Thermal Conditions group-box.

i. Specify the values for the parameters as per the following table:

Parameter	**Value**
Heat Transfer Coefficient (w/m2-k)	4
Free Stream Temperature (c)	20
Material Name	building-insulation
Wall Thickness (m)	0.1

(b) Click the Radiation tab.

Specify the values as per the radiant properties for walls mentioned in the table in Appendix.

i. Ensure that BC Type is set to opaque and Participates in Solar Ray Tracing is enabled.

ii. Enter 0.26 for Direct Visible and 0.9 for Direct IR in the Absorptivity group-box.

(c) Click OK to close the Wall panel.

9. Similarly set the boundary conditions for other internal walls namely w_east-wall, w_west-wall, w_room-walls, and w_pillars.

10. Set the boundary conditions for the remaining walls.

The remaining wall are w_ac-unit, w_door-top, w_door-top-shadow, w_glass-barriers, w_glass-barriers-shadow, w_landings, w_plants-and_furniture, w_south-wall, w_steel-frame-in-shadow, w_steel-frame-out-ends, w_steps, w_steps-shadow. *All these walls will have default thermal conditions (coupled or zero heat flux).*

(a) Retain the thermal condition for all the walls as default.

(b) Set the radiant properties according to the values provided in the table in Appendix.

The boundaries w_ac-unit *and* w_plants-and_furniture *will have radiant properties of* Furnishings*. The boundaries* w_door-top*,* w_door-top-shadow*, and* w_south-wall *will have radiant properties of* Walls*. The boundary* w_glass-barriers *will have radiant properties of* Internal Glass*. The boundaries* w_landings*,* w_steps*,* w_steps-shadow *will have radiant properties of* Flooring*. Keep default radiant properties for the rest.*

11. Set the boundary conditions for the air conditioning equipments.

    (a) Select Magnitude and Direction from the Velocity Specification Method drop-down list.

    (b) Enter `10` m/s for the Velocity Magnitude.

    (c) Enter `0.1, 1, 0` for X-Component of Flow Direction, Y-Component of Flow Direction and Z-Component of Flow Direction.

    (d) Select Intensity and Length Scale from the Specification Method drop-down list in the Turbulence group-box.

    (e) Enter `10` % for Turbulent Intensity and `0.02` m for Turbulent Length Scale.

    (f) Click the Thermal tab and enter `15` c for the Temperature.

    (g) Click OK to close the Velocity Inlet panel.

12. Close the Boundary Conditions panel.

Step 6: Solution

1. Define the solution control parameters.

   Solve ⟶ Controls ⟶ Solution...

   (a) Enter `0.3` for Pressure, `0.2` for Momentum and `0.9` for Energy in the Under-Relaxation Factors group-box.

   You can also solve this problem with the default relaxation factors initially. This will provide optimum convergence rates if they are suitably stable. If larger relaxation is required then the suggested values for momentum and energy are 0.3 to 0.5 and 0.8 to 0.9 respectively.

   (b) Select Body Force Weighted as a discretization scheme for Pressure and retain default First Order Upwind discretization scheme for other parameters.

   Natural convection problems are inherently unstable and it is better to solve them using First Order Upwind *discretization scheme in the beginning. When convergence is achieved, higher order schemes can be used.*

   (c) Click OK to close Solution Controls panel.

2. Define surface monitor.

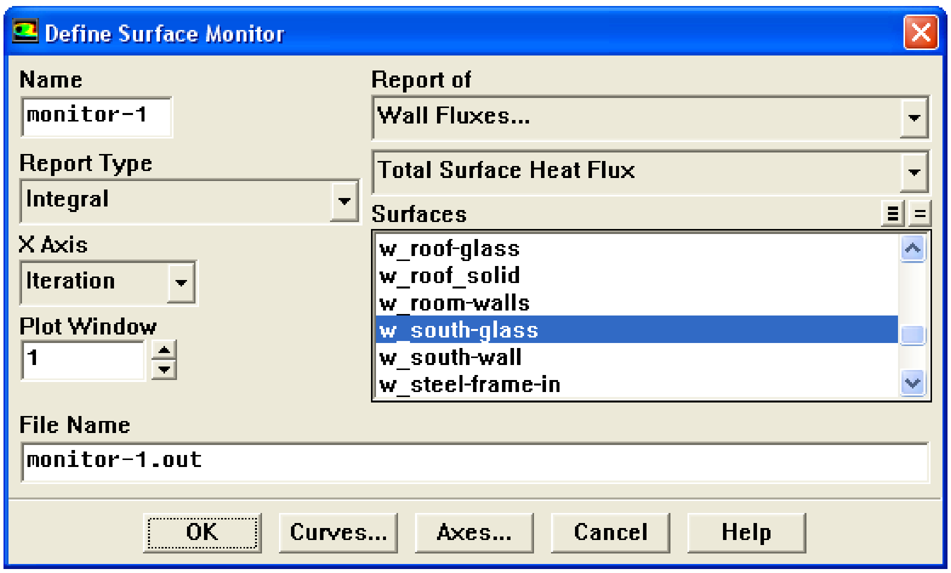
   Solve ⟶ Monitors ⟶ Surface...

   Due to high transience it may be difficult to achieve good convergence in this case, it is useful to observe the solution progress by monitoring some useful surfaces. Here, you will monitor the heat transfer across the glass.

   (a) Enter 1 for the Surface Monitors.

   (b) Enable Plot, Print, and Write options for monitor-1 .

   (c) Click the Define... button to open the Define Surface Monitor panel.



   i. Select Wall Fluxes... and Total Surface Heat Flux from the Report of drop-down lists.

   ii. Select w_southglass from the Surfaces list.

   iii. Set 1 for the Plot Window.

   iv. Click OK to close the Define Surface Monitor panel.

   (d) Click OK to close the Surface Monitors panel.

3. Initialize the solution.

   Solve ⟶ Initialize ⟶ Initialize...

   (a) Select all-zones from the Compute From drop-down list.

   (b) Enter 22 for Temperature and 0 for all the remaining parameters in the Initial Values group-box.

   (c) Click Init and close the Solution Initialization panel.

   Initialization may take a little more time than usual as the solar load model calculates the loads to be applied to all surfaces.

4. Save the case and data files (`fel_atrium.cas` and `fel_atrium.dat`).

   File ⟶ Write ⟶ Case & Data...

   The postprocessing tools can be used at this stage to review the solar load that each surface will be subject to. This can easily be done by plotting contours of selecting any of the following wall fluxes:

   - Solar Heat Flux
   - Absorbed Visible Solar Flux
   - Absorbed IR Solar Flux
   - Reflected Visible Solar Flux
   - Reflected IR Solar Flux
   - Transmitted Visible Solar Flux
   - Transmitted IR Solar Flux

5. Iterate the solution for 1000 iterations.

   Solve ⟶ Iterate...

6. Save the case and data files (`fel_atrium-1.cas` and `fel_atrium-1.dat`).

   File ⟶ Write ⟶ Case & Data...

Step 7: Include Radiation Model

Activate a radiation model to include the radiant heat exchange between the internal surfaces. This is needed because after inspecting the initial results you will see high temperatures, particularly on the second floor.

1. Activate P1 or S2S radiation model.

   Both models will run with a comparable speed, but the S2S model will take several hours to establish the view factors. The results from the S2S model will be more accurate.

   Define ⟶ Models ⟶ Radiation...

2. In the Boundary Conditions panel, set the Internal Emissivity for all the Wall boundaries to the value same as that of Direct IR absorptivity (α_{IR}).

   Define ⟶ Boundary Conditions...

3. If you are using S2S model, do the following settings in the Radiation Model panel.

   Define ⟶ Models ⟶ Radiation...

   (a) Under Iteration Parameters, reduce the value of Flow Iterations per Radiation Iteration to 5.

   (b) Under Parameters, click Set....

      i. Set Faces per Surface Cluster for Flow Boundary Zones to 10.

      ii. Click Apply to All Walls and close the panel.

4. Iterate the solution for 1000 more iterations.

   Solve ⟶Iterate...

5. Save the case and data files (`fel_atrium-p1.cas` and `fel_atrium-p1.dat`).

   File ⟶ Write ⟶Case & Data...

Step 8: Postprocessing

1. Create isosurfaces at x=3.5 and y=1.

   Surface ⟶Iso-Surface...

2. Display contours of Static Temperature on the ground floor at y=1 (Figure 3).

   Display ⟶Contours...

3. Display contours of Static Temperature at x=3.5 (Figure 4).

4. Display contours of Solar Heat Flux on w_south-glass (Figure 5).

5. Display contours of Transmitted Visible Solar Flux on the glass surfaces (Figure 6).

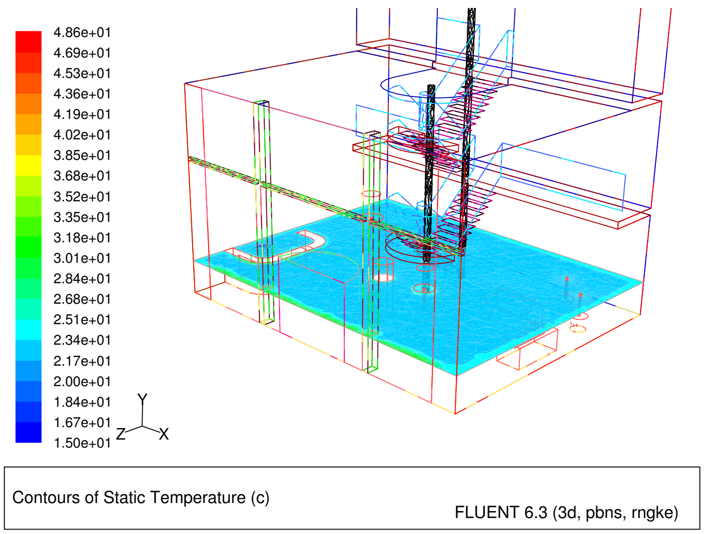


Figure 3: Contours of Static Temperature at y=1

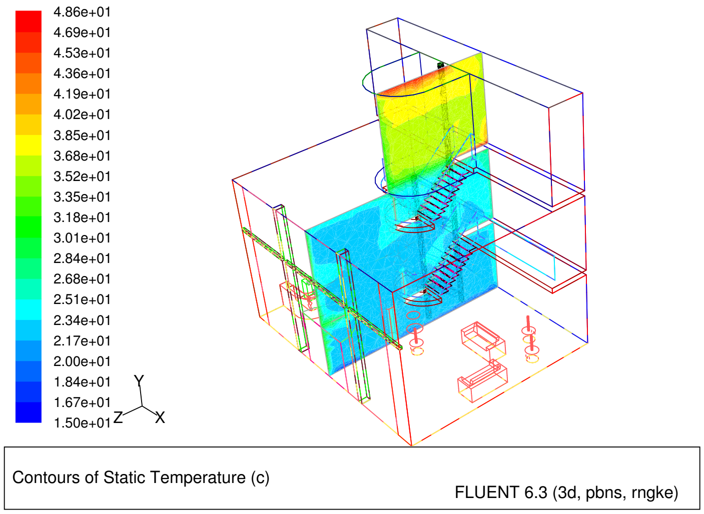


Figure 4: Contours of Static Temperature at x=3.5

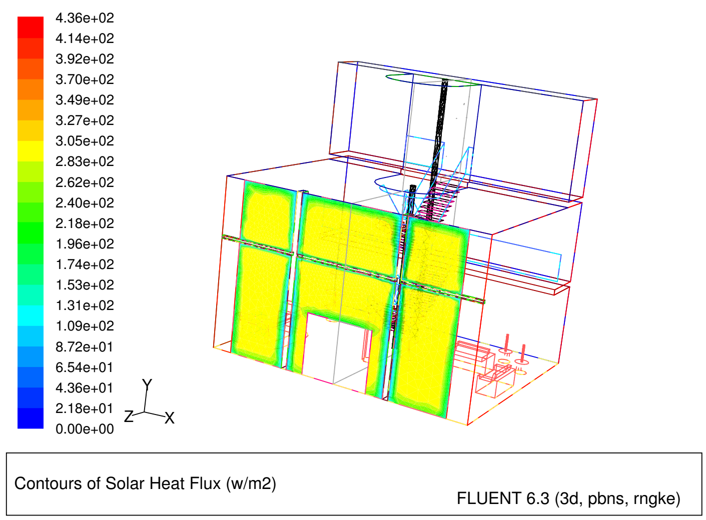


Figure 5: Contours of Solar Heat Flux on w_south-glass

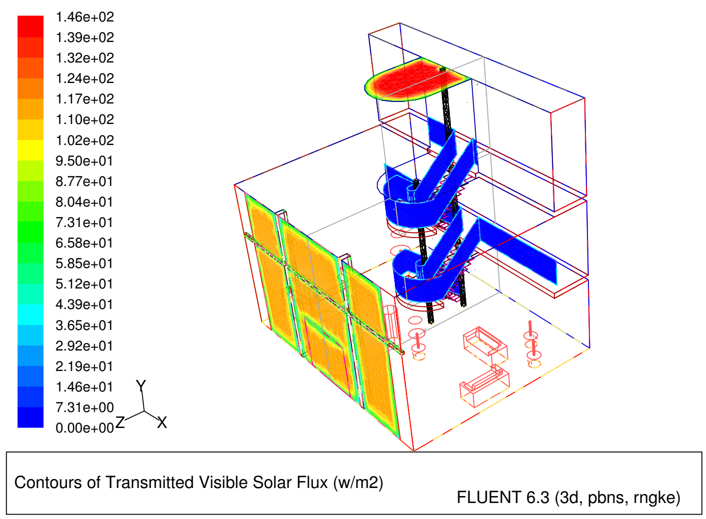


Figure 6: Contours of Transmitted Visible Solar Flux on the Glass Surfaces

Appendix

For all opaque materials, you need to know the absorptivity for the infrared (long wave) and visible (short wave) bands. Generally, absorptivity is higher in the infrared band. Reliable inputs for this data may be difficult to obtain from manufacturers and/or suppliers. So some estimation may be required, based on the data provided in standard heat transfer textbooks.

For transparent surfaces you need to provide the absorptivity and transmissivity for infra-red and visible components of direct radiation. The value you specify should be for normal incident radiation. FLUENT will automatically adjust for the actual angle of incidence. You also need to provide the overall absorptivity and transmissivity for diffuse (mainly infra-red) radiation. This will be the hemispherically averaged value.

If you have difficulty obtaining values for glass, refer to following table (*Table 13 in Chapter 30 of the 2001 ASHRAE Fundamentals Handbook*):

Surface	**Material**	**Radiant Properties**
Walls	Matt White Paint	$\alpha_V = 0.26$, $\alpha_{IR} = 0.9$
Flooring	Dark Grey Carpet	$\alpha_V = 0.81$, $\alpha_{IR} = 0.92$
Furnishings	Various, generally mid coloured matt	$\alpha_V = 0.75$, $\alpha_{IR} = 0.90$
Steel Frame	Dark gray gloss	$\alpha_V = 0.78$, $\alpha_{IR} = 0.91$
External Glass	Double glazed coated glass	$\alpha_V = 0.49$, $\alpha_{IR} = 0.49$, $\alpha_D = 0.49$, $\tau_V = 0.3$, $\tau_{IR} = 0.3$, $\tau_D = 0.32$
Internal Glass	Single layer clear float glass	$\alpha_V = 0.09$, $\alpha_{IR} = 0.09$, $\alpha_D = 0.1$, $\tau_V = 0.83$, $\tau_{IR} = 0.83$, $\tau_D = 0.75$