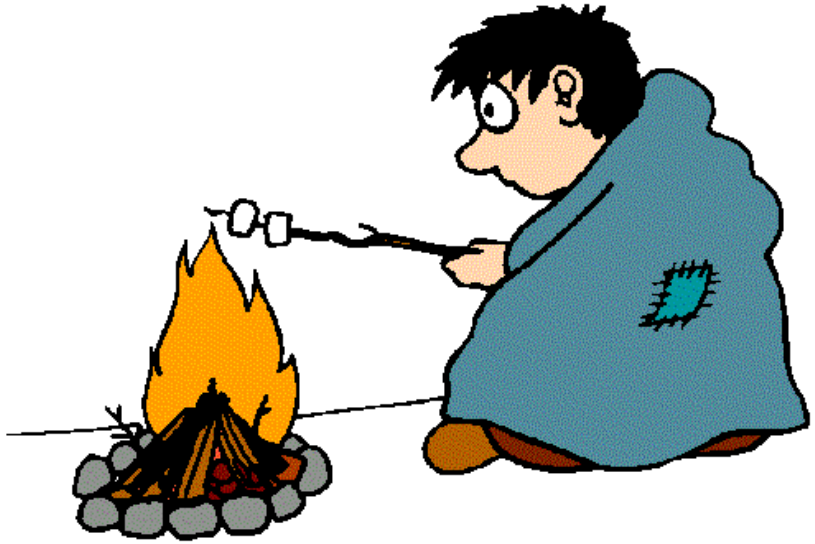
Radiation in ANSYS FLUENT



ANSYS® FLUENT®

Overview

- ANSYS Fluent provides five radiation models that allow us to include radiation, with or without a participating medium, in heat transfer simulations.
- Heating or cooling of surfaces due to radiation and/or heat sources or sinks due to radiation within the fluid phase can be included in your model using one of the following radiation models.

- Discrete Transfer Radiation Model (DTRM)
- P1 Radiation Model
- Rosseland Model
- Surface-to-Surface (S2S)
- Discrete Ordinates Model (DOM)

In addition to these radiation models, ANSYS Fluent also provides a solar load model that allows us to include the effects of solar radiation in the simulations.

Applications

We should include radiative heat transfer in our simulations when the radiant heat flux is large compared to the heat transfer rate due to convection or conduction.

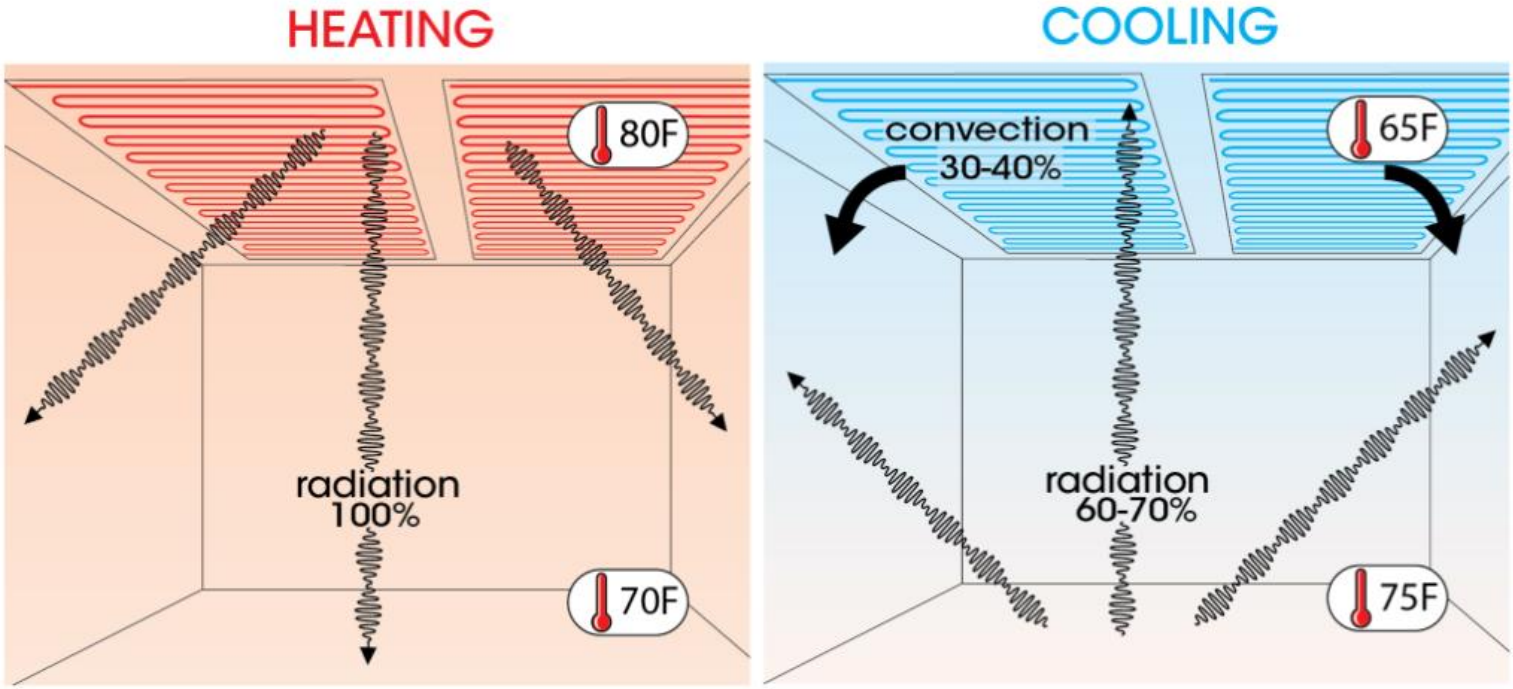
$$Q_{rad} = \sigma\left(T_{max}^4 - T_{min}^4\right)$$

Applications

Typical applications

- radiative heat transfer from flames
- surface-to-surface radiant heating or cooling
- coupled radiation, convection, and/or conduction heat transfer
- radiation through windows in HVAC (Heating, ventilation, and air conditioning) applications, and cabin heat transfer analysis in automotive and aircraft applications
- radiation in glass processing, glass fiber drawing, and ceramic processing (materials processing)

Applications



how radiant ceiling heating and cooling work

Radiative Transfer Equation

The radiative transfer equation (RTE) for an absorbing, emitting, and scattering medium at position $\vec{r}$ in the direction $\vec{s}$ is

$$\frac{dI(\vec{r},\vec{s})}{ds}+(a+\sigma_s)I(\vec{r},\vec{s})=an^2\frac{\sigma T^4}{\pi}+\frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\vec{r},\vec{s}')\Phi(\vec{s}\cdot\vec{s}')d\Omega'$$

$\vec{r}$ = position vector

$\vec{s}$ = direction vector

$\vec{s}'$ = scattering direction vector

s = path length

a = absorption coefficient

n = refractive index

σ_s = scattering coefficient

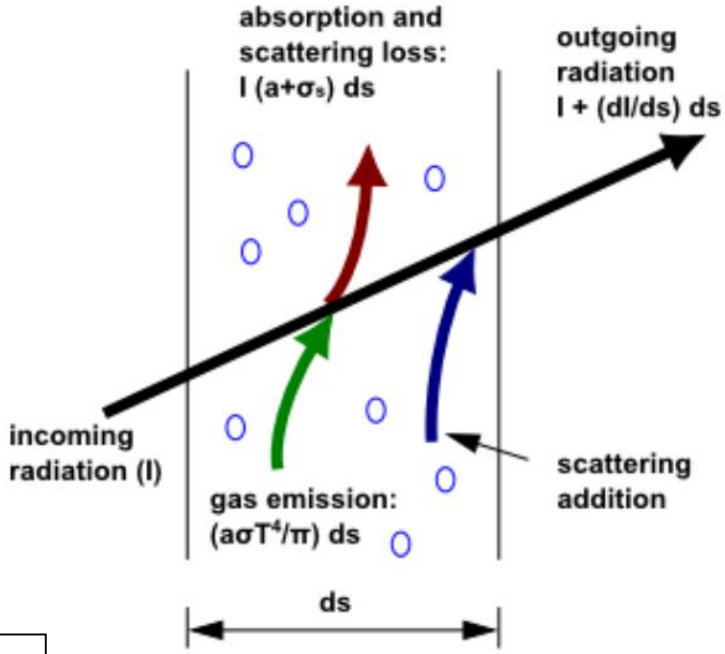
σ = Stefan-Boltzmann constant ($5.669 \times 10^{-8}\ W/m^2\text{-}K^4$)

I = radiation intensity, which depends on position $(\vec{r})$ and direction $(\vec{s})$

T = local temperature

Φ = phase function

Ω' = solid angle



$a+\sigma_s$ is extinction coefficient of the medium. The refractive index n is important when considering radiation in semi-transparent media.

Radiative Heat Transfer

1. Discrete Transfer Radiation Model

- ✓ **Main assumption – Radiation leaving a surface element within a specified range of solid angles can be approximated by a single ray.**
- ✓ **Uses a ray-tracing technique to integrate radiant intensity along each ray:**

$$\frac{dI}{ds} + a\,I = \frac{a\,\sigma\,T^4}{\pi}$$

✓ **Advantages:**

- **Relatively simple model.**
- **Can increase accuracy by increasing number of rays.**
- **Applicable to a wide range of optical thicknesses.**

✓ **‹Limitations:**

- **Assumes all surfaces are diffuse.**
- **Effect of scattering not included.**
- **Solving a problem with a large number of rays is CPU-intensive.**

2. P-1 model

Main assumption – The directional dependence in RTE is integrated out, resulting in a diffusion equation for incident radiation.

✓ **Advantages:**

- **Radiative transfer equation easy to solve with small CPU demand.**
- **Includes effect of scattering. Thus effects of particles, droplets, and soot can be included.**
- **Works reasonably well for applications where the optical thickness is large (e.g., combustion).**

✓ **Limitations:**

- **Assumes all surfaces are diffuse.**
- **May result in loss of accuracy (depending on the complexity of the geometry) if the optical thickness is small.**
- **Tends to overpredict radiative fluxes from localized heat sources or sinks.**
- **Boundary conditions might be tricky.**

3. Rosseland Model

- ✓ **Advantages:**

  The Rosseland model has two advantages over the P-1 model. Since it does not solve an extra transport equation for the incident radiation (as the P-1 model does), the Rosseland model is faster than the P-1 model and requires less memory.

- ✓ **Limitations:**

  The Rosseland model can be used only for optically thick media. Note also that the Rosseland model is not available when the density-based solver is being used; it is available with the pressure-based solver, only.

4. Surface-to-Surface Radiation Model

The S2S radiation model can be used for modeling radiation in situations where there is no participating media.

- **For example, spacecraft heat rejection system, solar collector systems, radiative space heaters, and automotive underhood cooling.**
- **S2S is a view-factor based model.**
- **Non-participating media is assumed.**

Limitations:

- **The S2S model assumes that all surfaces are diffuse.**
- **The implementation assumes gray radiation.**
- **Storage and memory requirements increase very rapidly as the number of surface faces increases. Memory requirements can be reduced by using clusters of surface faces.**
- **Clustering does not work with sliding meshes or hanging nodes.**
- **Not to be used with periodic or symmetry boundary conditions.**

5. Discrete Ordinates Model

The radiative transfer equation is solved for a discrete number of finite solid angles, σ_s:

$$\frac{\partial I}{\partial x_i} + \underbrace{(a+\sigma_s)\, I(r,s)}_{\text{Absorption}} = \underbrace{a\, n^2 \frac{\sigma T^4}{\pi}}_{\text{Emission}} + \underbrace{\frac{\sigma_s}{4\pi}\int_0^{4\pi} I(r,s')\,\Phi(s\cdot s')\, d\Omega'}_{\text{Scattering}}$$

✓ **Advantages:**

- **Conservative method leads to heat balance for coarse discretization. Accuracy can be increased by using a finer discretization.**
- **Most comprehensive radiation model: Accounts for scattering, semi-transparent media, specular surfaces, and wavelength-dependent transmission using banded-gray option.**

√ **Limitations**

- **Solving a problem with a large number of ordinates is CPU-intensive.**

Gray radiation and non-gray radiation

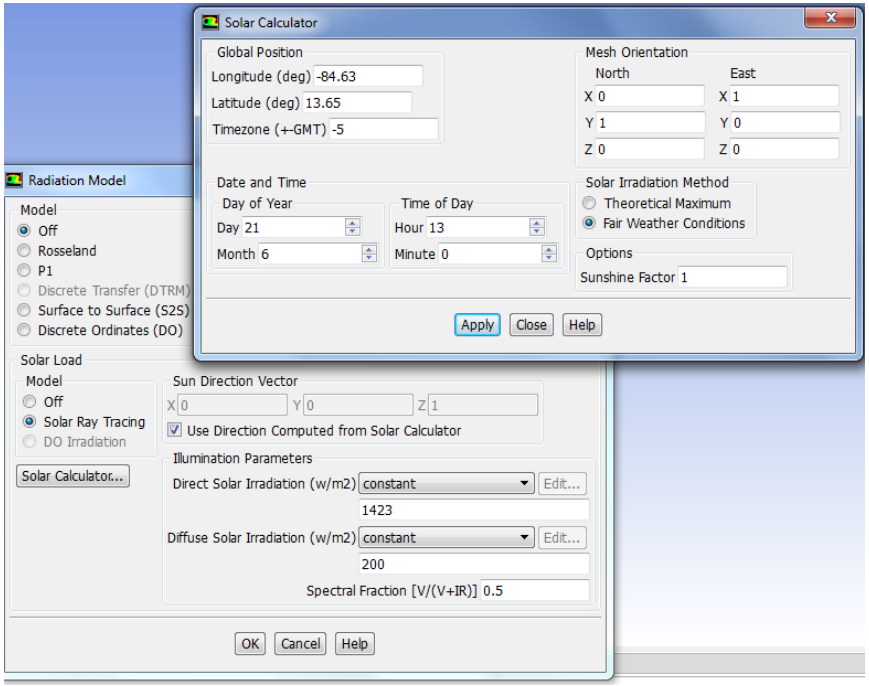
A gray surface has a spectral emissivity and absorptivity that is not dependent on the wavelength of the radiation. Gray radiation refers to the type of radiation one would expect to see from a gray surface.

The gray model offers a huge simplification to real problems. The gray model should always be used as a first option wherever possible. If more accurate solutions are desired, then it may be appropriate to use a non-gray model.

Solar Load Model

- ✓ **Solar load model**
- ▪ **Ray tracing algorithm for solar radiant energy transport: Compatible with all radiation models**
- ▪ **Available with parallel solver (but ray tracing algorithm is not parallelized)**
- ▪ **3D only**

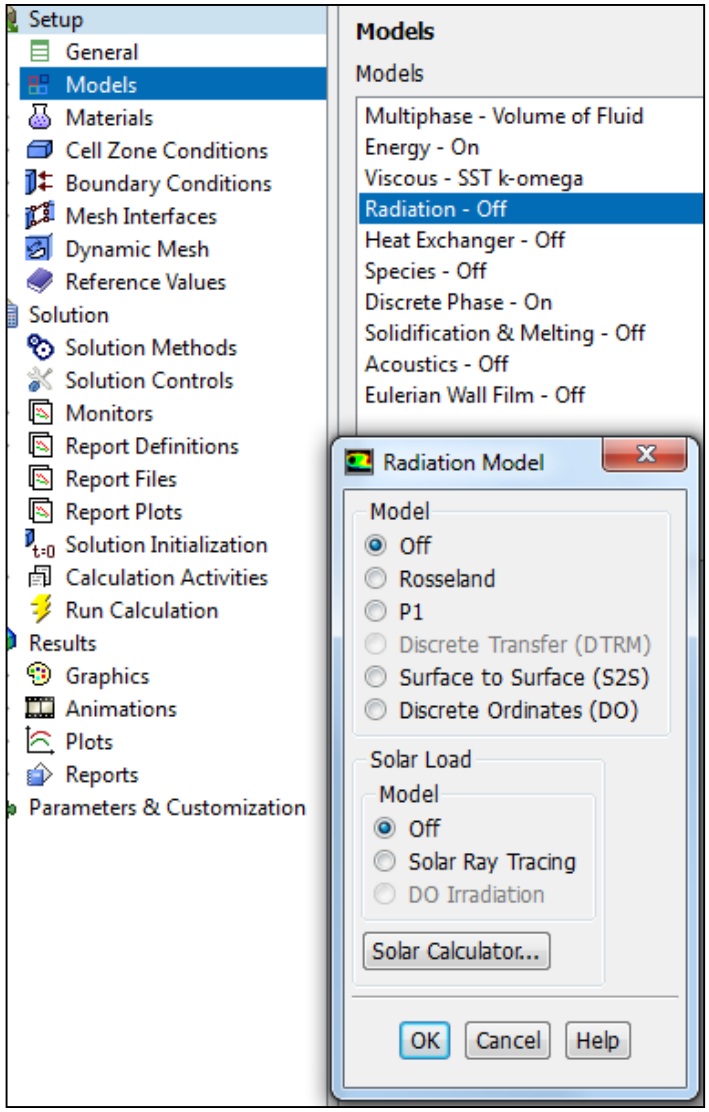
- ✓ **Specifications**
- ▪ **Sun direction vector**
- ▪ **Solar intensity (direct, diffuse)**
- ▪ **Solar calculator for calculating direction and direct intensity using theoretical maximum or "fair weather conditions"**
- ▪ **Transient cases**



Choosing a radiation model

For certain problems, one radiation model may be more appropriate than others.

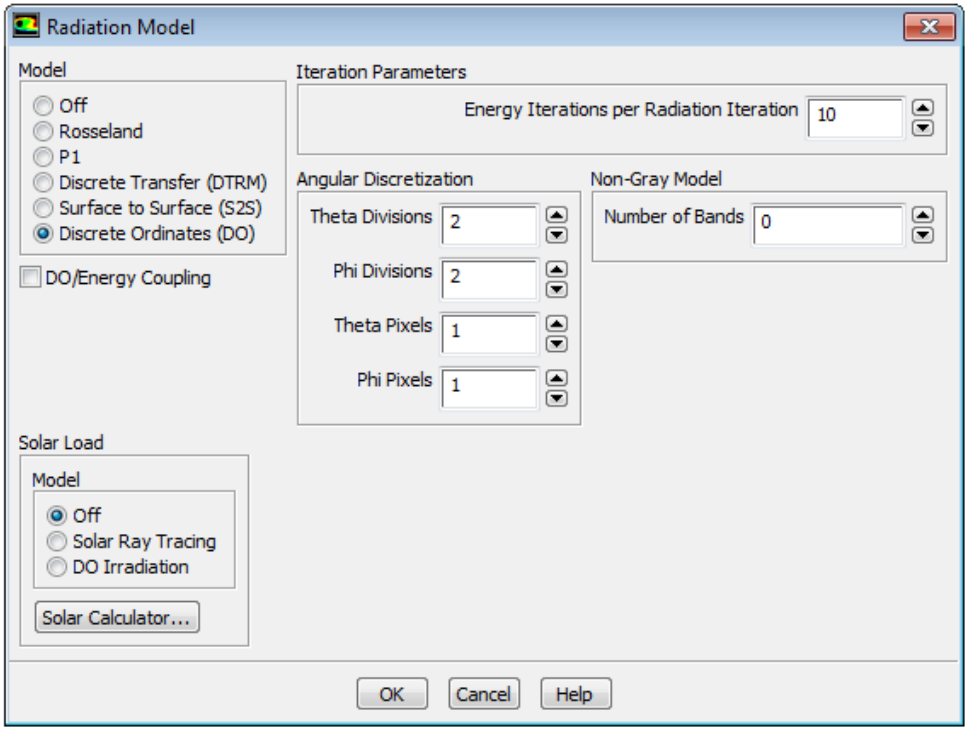
- Computational effort – **P1** gives reasonable accuracy with less effort.
- Accuracy – **DTRM and DOM** more accurate.
- Optical thickness – **DTRM/DOM** for optically thin media (αL << small); **P1** better for optically thick media.
- Scattering – **P1 and DOM** account for scattering.
- Particulate effects – **P1 and DOM** account for radiation exchange between gas and particulates.
- Localized heat sources – **DTRM/DOM** with sufficiently large number of rays/ordinates is more appropriate.



Main steps

The procedure for setting up and solving a radiation problem is outlined below, and described in detail in referenced sections. Steps that are relevant only for a particular radiation model are noted as such. The steps that are pertinent to radiation modeling, are shown here.

Setup → Models → Radiation → Edit...



Related settings

➢ **Set the appropriate radiation parameters.**

- **If we are modeling non-gray radiation using the P-1 model, define the non-gray radiation parameters as described in *Setting Up the P-1 Model with Non-Gray Radiation*.**
- **If we are using the DTRM, define the ray tracing as described in *Setting Up the DTRM*.**
- **If we are using the S2S model, define the surface clusters and view factors settings and compute or read the view factors as described in *Setting Up the S2S Model*.**
- **If you are using the DO model, choose DO/Energy Coupling if desired, define the angular discretization as described in *Setting Up the DO Model* and, if relevant, define the non-gray radiation parameters as described in *Defining Non-Gray Radiation for the DO Model*.**

Related settings

- **Define the material properties as described in *Defining Material Properties for Radiation*.**
- **Define the boundary conditions as described in *Defining Boundary Conditions for Radiation*. If the model contains a semi-transparent medium, see the information below on setting up semi-transparent media.**
- **Set the parameters that control the solution (DTRM, DO, S2S, and P-1 only) as described in *Solution Strategies for Radiation Modeling*.**
- **Run the solution as described in *Running the Calculation*.**
- **Postprocess the results as described in *Postprocessing Radiation Quantities*.**

Example using Fluent

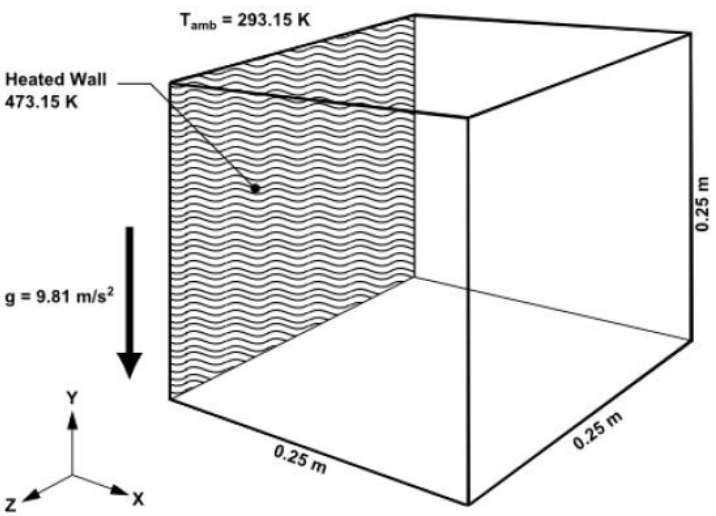
In this example, combined radiation and natural convection are solved in a three-dimensional square box on a mesh consisting of hexahedral elements.

This example demonstrates how to do the following:

- Use the surface-to-surface (S2S) radiation model in ANSYS FLUENT.
- Set the boundary conditions for a heat transfer problem involving natural convection and radiation.
- Calculate a solution using the pressure-based solver.
- Display velocity vectors and contours of wall temperature, surface cluster ID, and radiation heat flux.

Problem Description

A three-dimensional box 0.25 m×0.25 m ×0.25 m has a hot wall of aluminum at 473.15 K. All other walls are made of an insulation material and are subject to radiative and convective heat transfer to the surrounding environment, which is at 293.15 K. Gravity acts downwards. The medium contained in the box is assumed not to emit, absorb, or scatter radiation. All walls are gray. The objective is to compute the flow and temperature patterns in the box, as well as the wall heat flux, using the surface-to-surface (S2S) model available in ANSYS FLUENT.



The working fluid has a Prandtl number of approximately 0.71, and the Rayleigh number based on L (0.25m) is 10^8. This means the flow is most likely laminar. The Planck number is 0.006, and measures the relative importance of conduction to radiation.

Main steps

Step 1: Mesh

Step 2: General Settings

Step 3: Models

Step 4: Materials

Step 5: Boundary Conditions

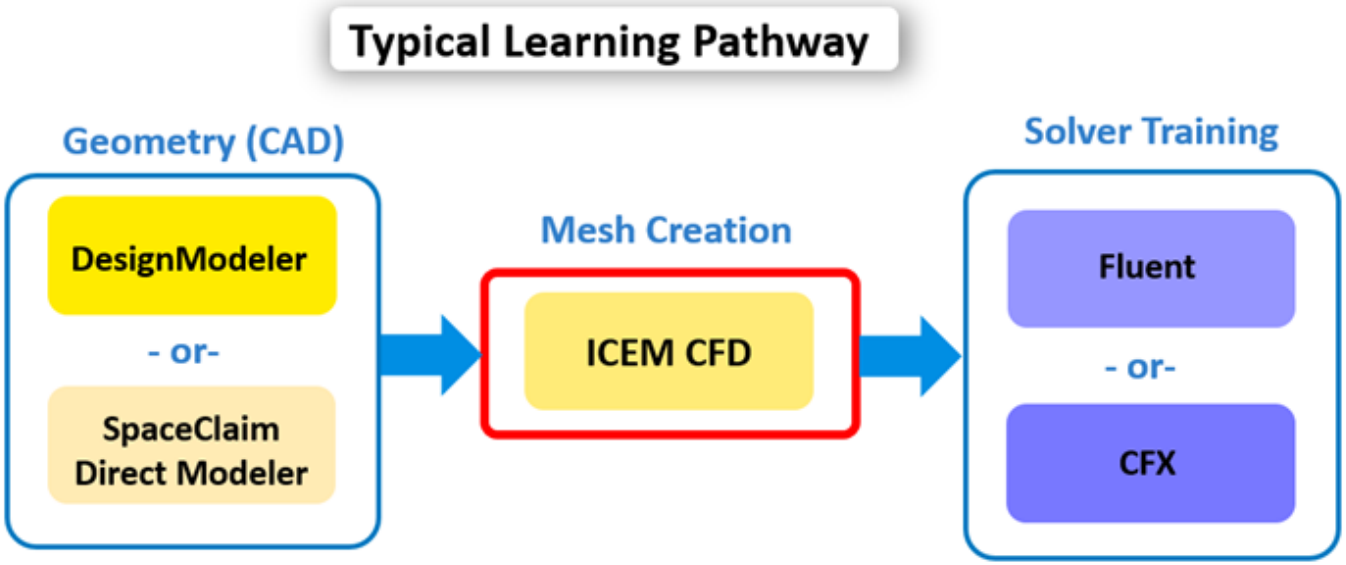
Step 6: Solution

Step 7: Postprocessing

Step 8: Compare the Contour Plots after Varying Radiating Surfaces

Step 9: S2S Definition, Solution, and Postprocessing with Partial Enclosure

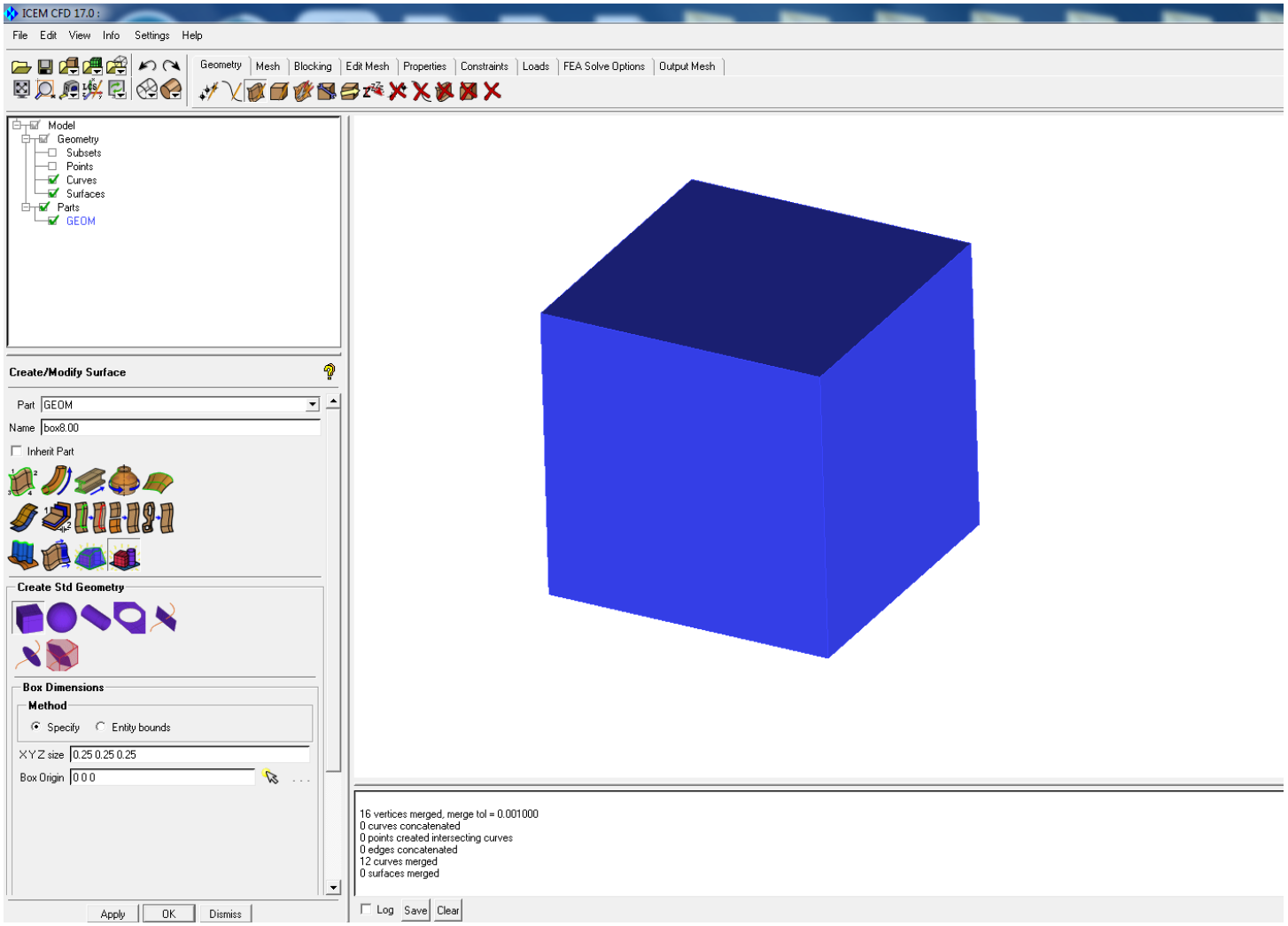
Step 1: Mesh



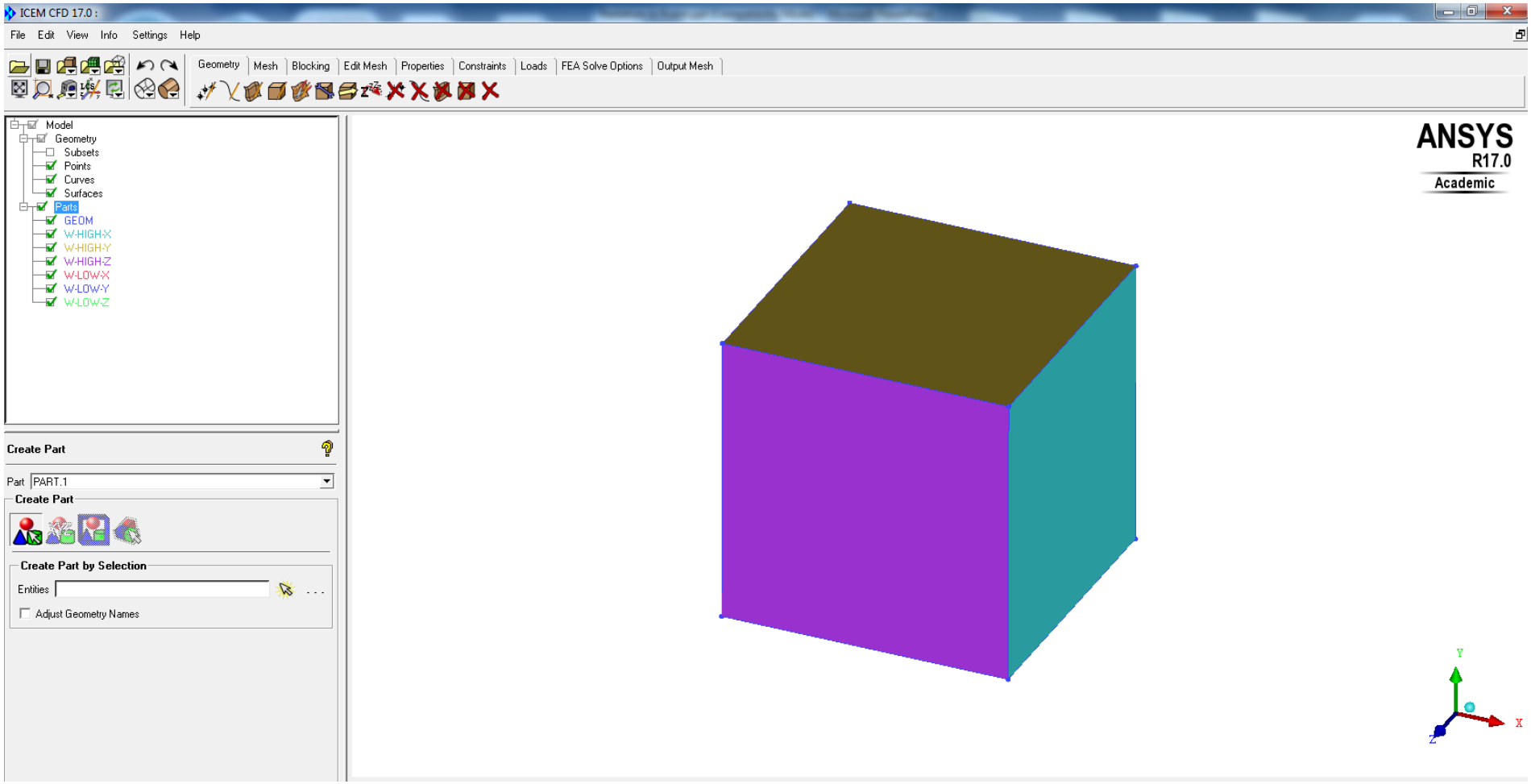
ANSYS ICEM CFD

Step 1: Mesh-geometry

Geometry>surface>standard_shapes

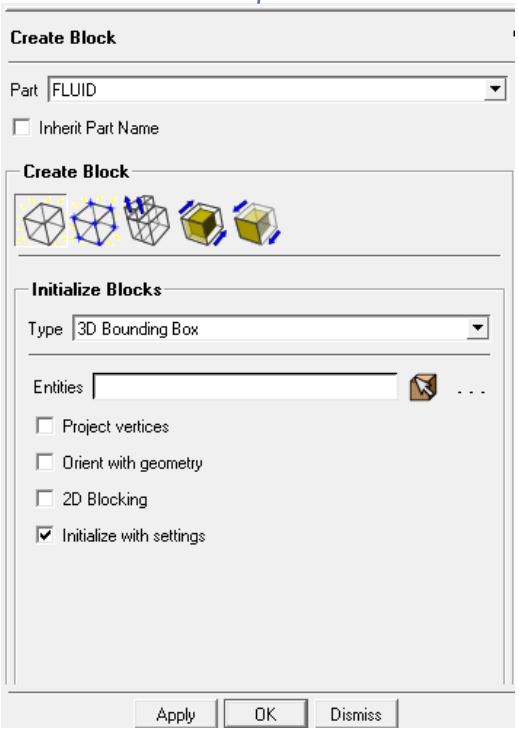


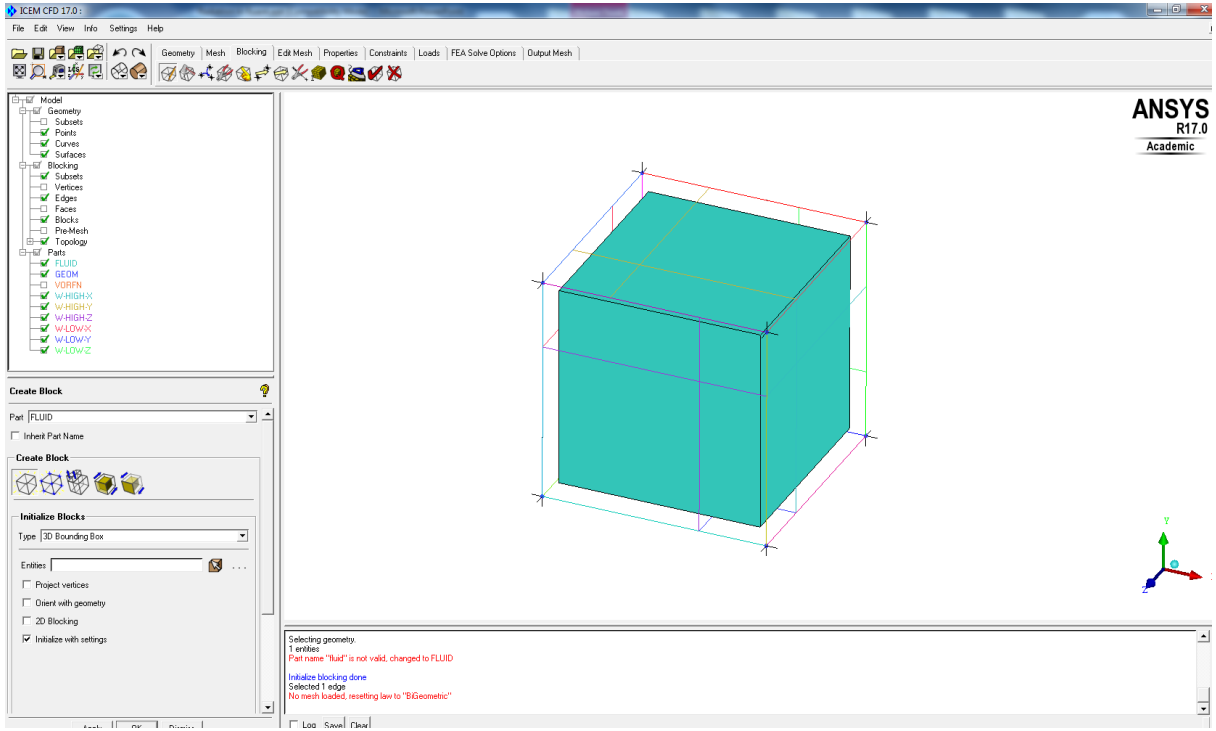
Step 1: Mesh-creating part



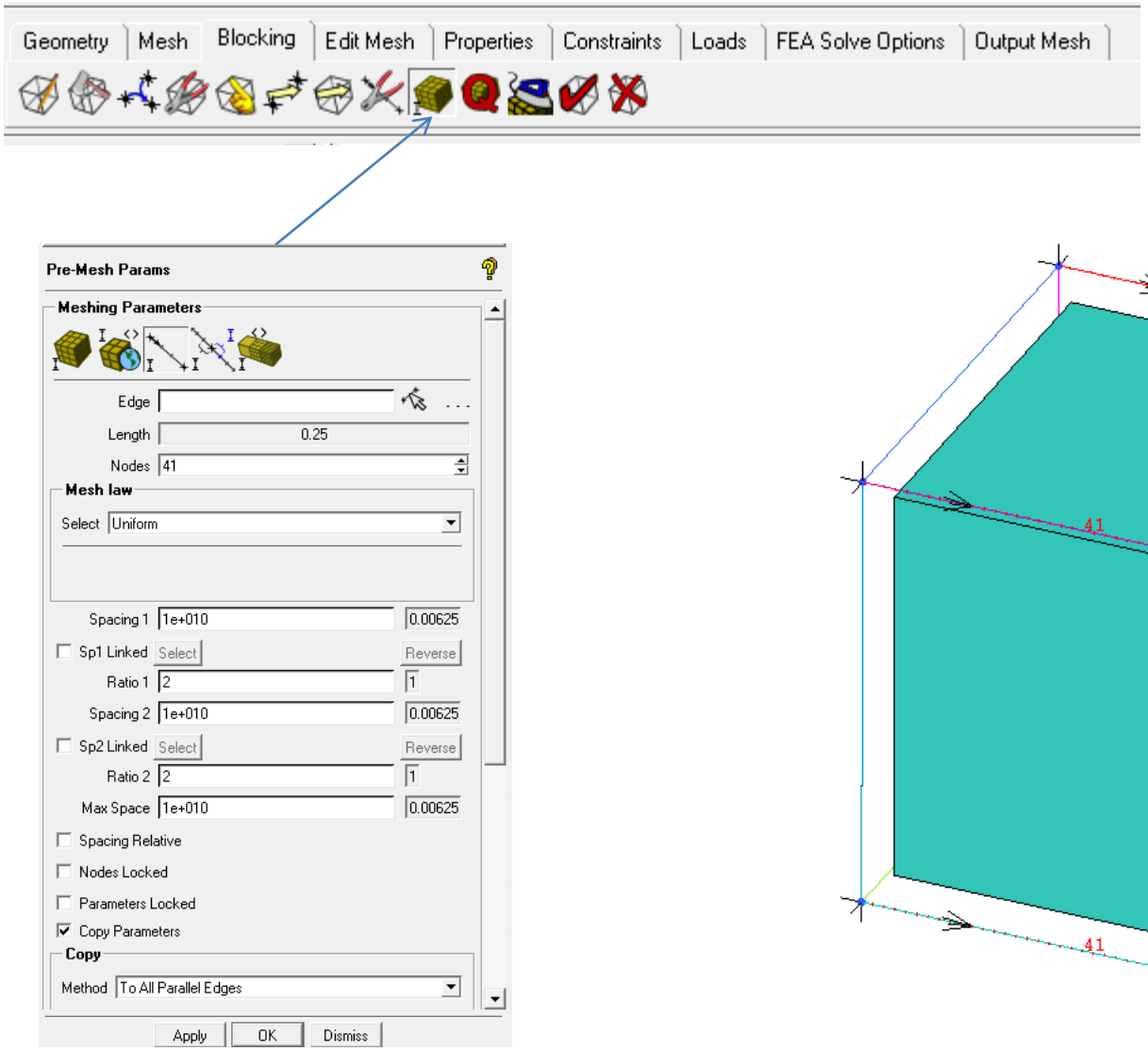
Step 1: Mesh-creating block

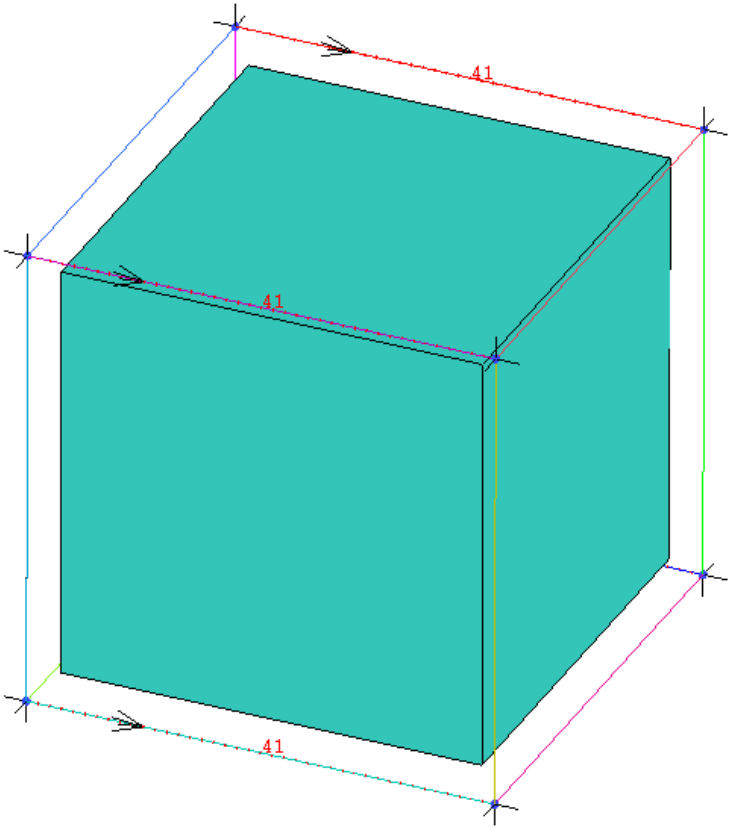
Structured mesh



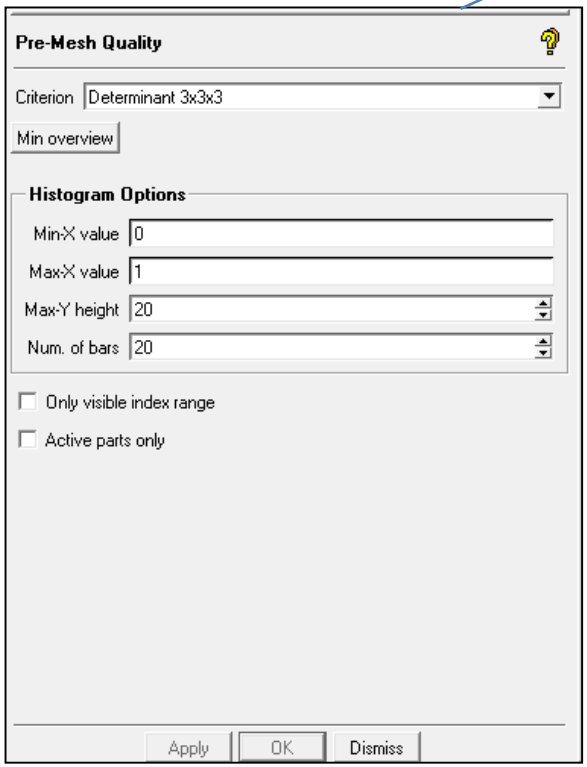


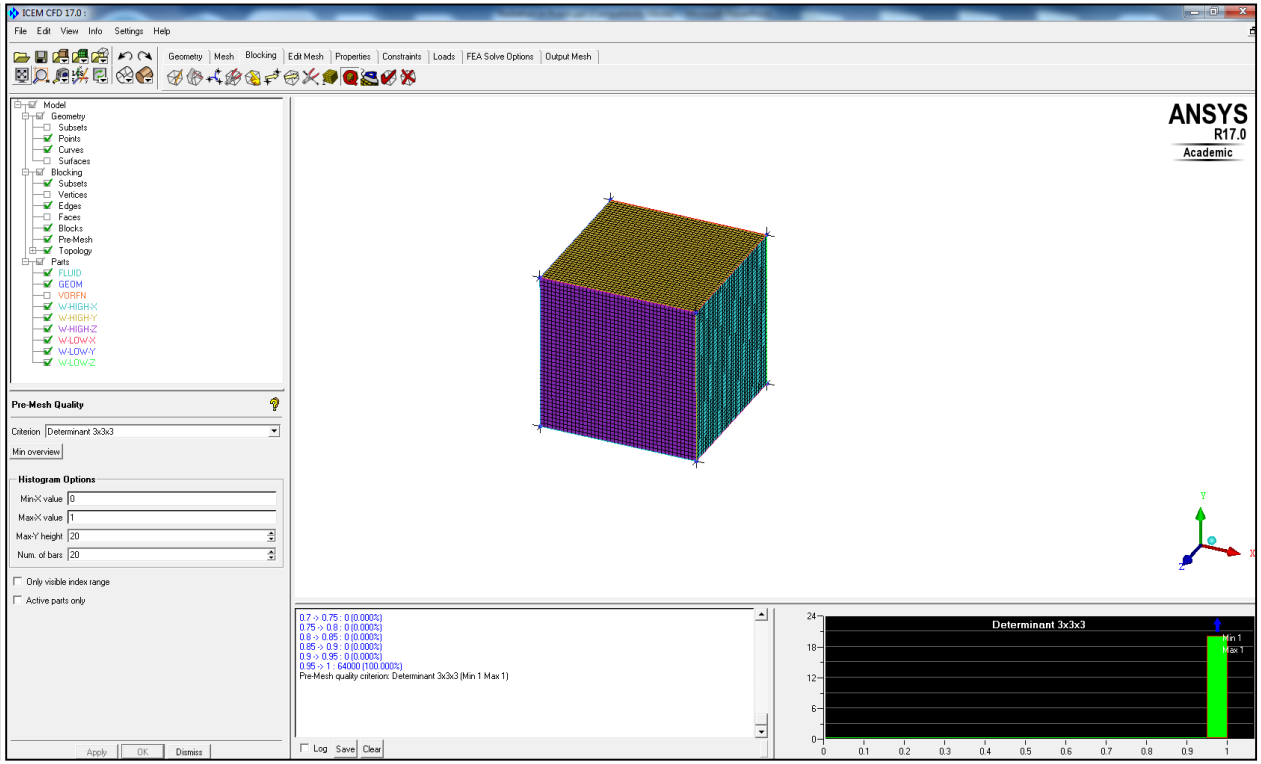
Step 1: Mesh-setting nodes



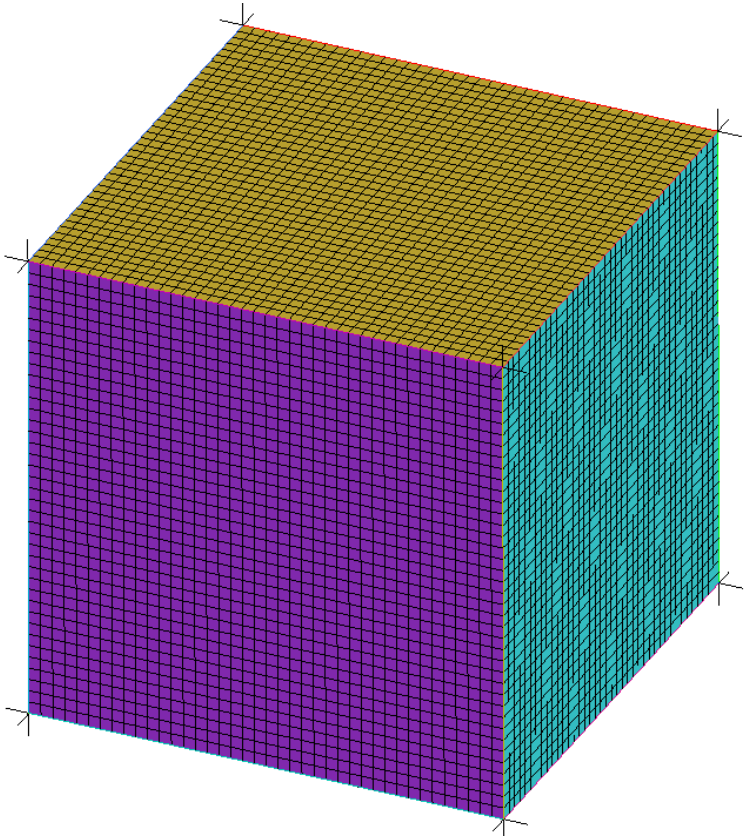


Step 1: Mesh-check mesh quality

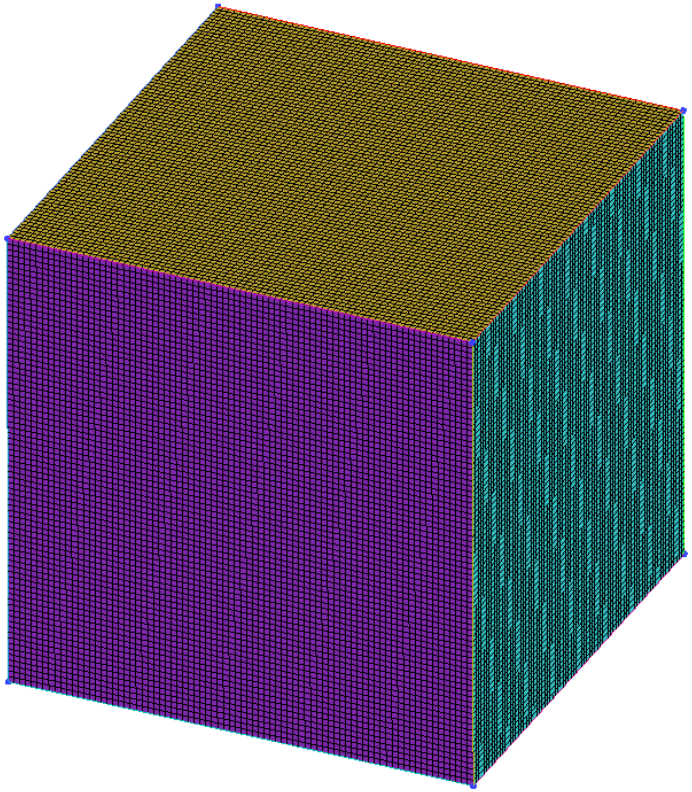




Step 1: Mesh

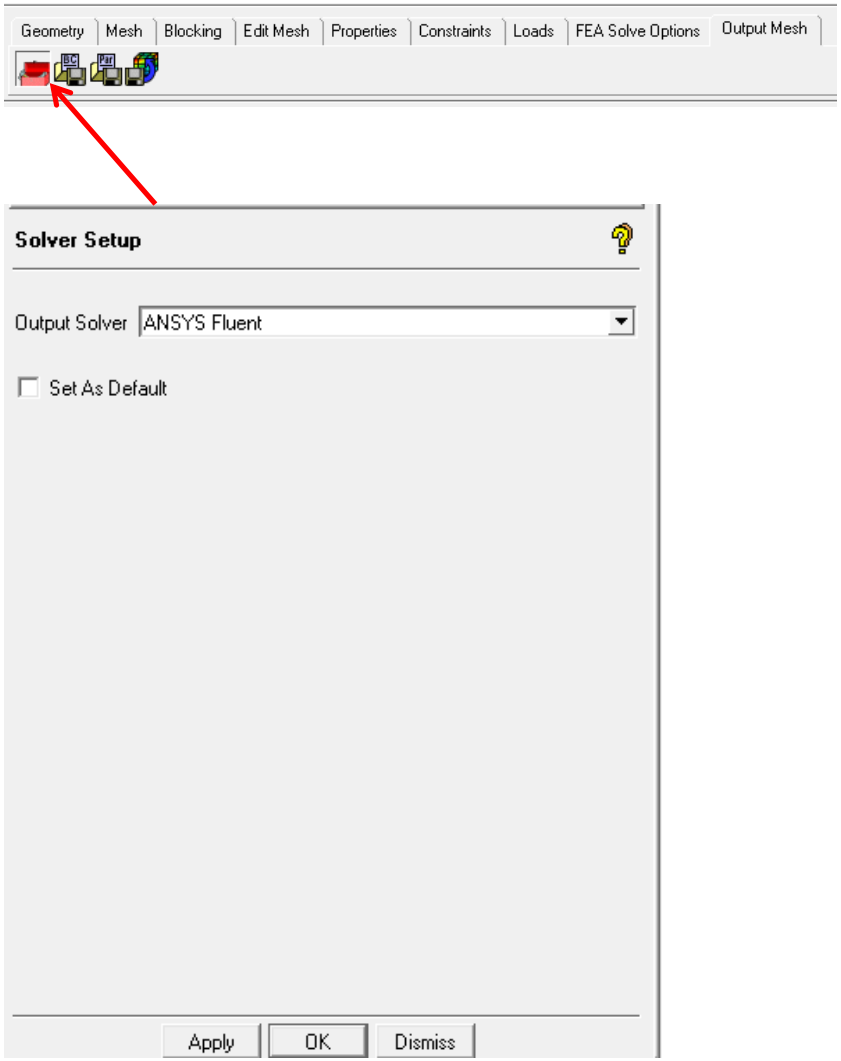


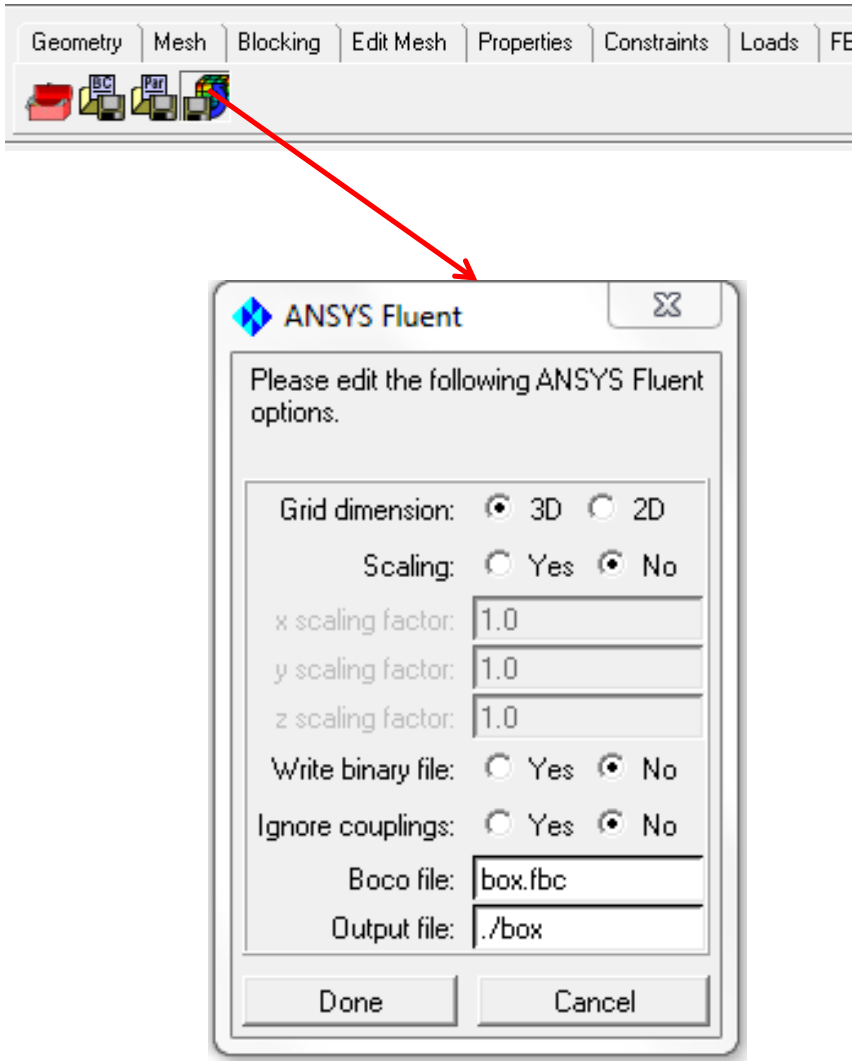
(40×40×40)



(80×80×80)

Step 1: Mesh-output mesh





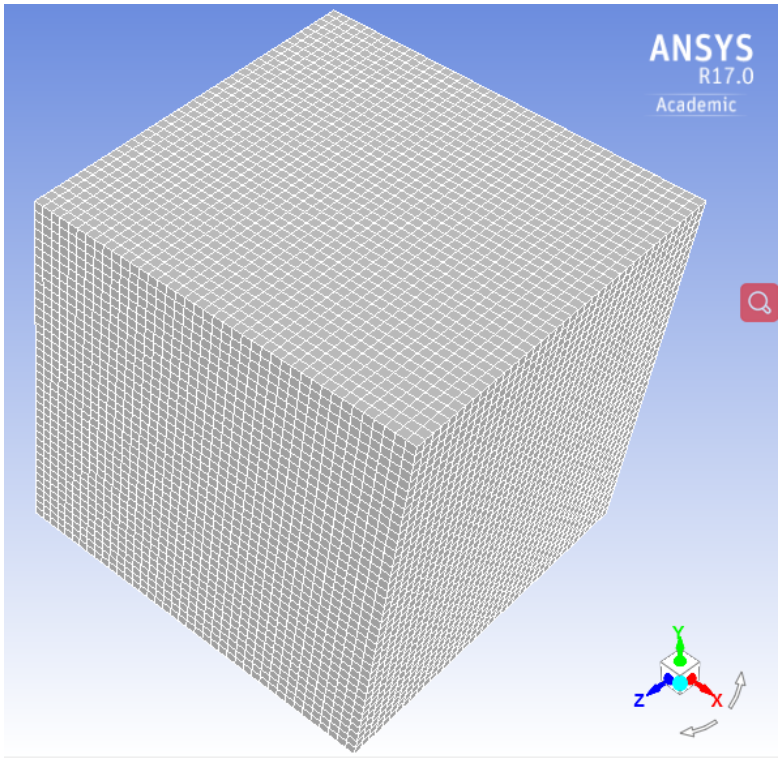
Step 1: Mesh

The mesh size will be reported as 64,000 cells (40×40×40).

Generated by Software ICEM. Using Fluent 17.0 read the mesh.

Check the mesh.

ANSYS FLUENT will perform various checks on the mesh and report the progress in the console. Make sure that the reported minimum volume is a positive number.

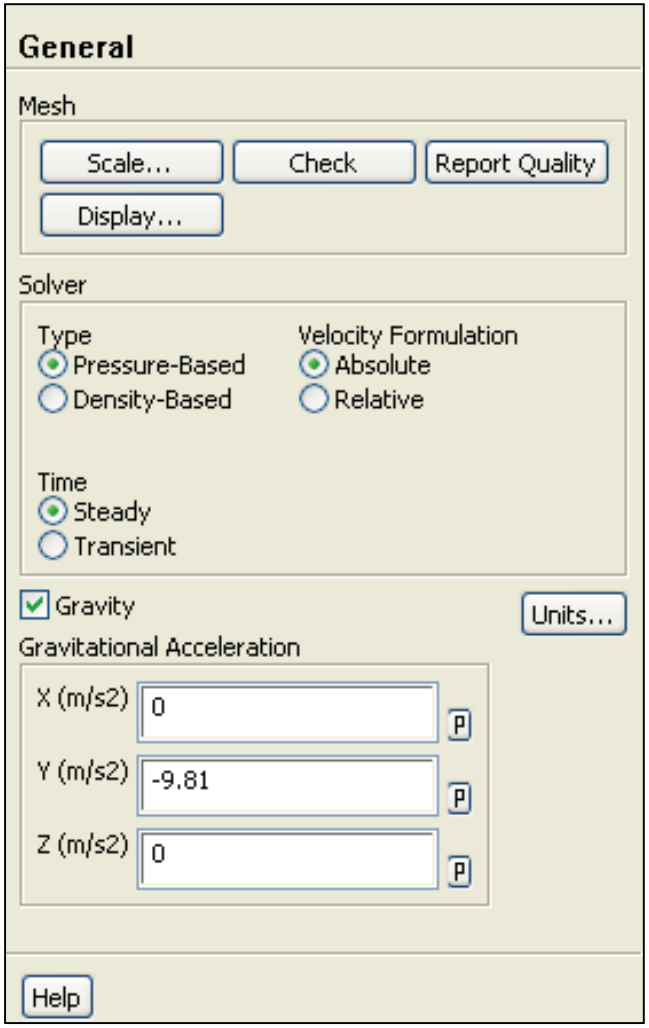


Step 2: General Settings

a. Retain the default settings in the Solver group box.

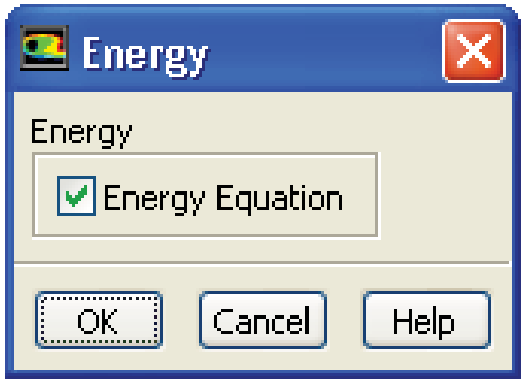
b. Enable the Gravity option.

c. Enter -9.81 m/s^2 for Y in the Gravitational Acceleration group box.

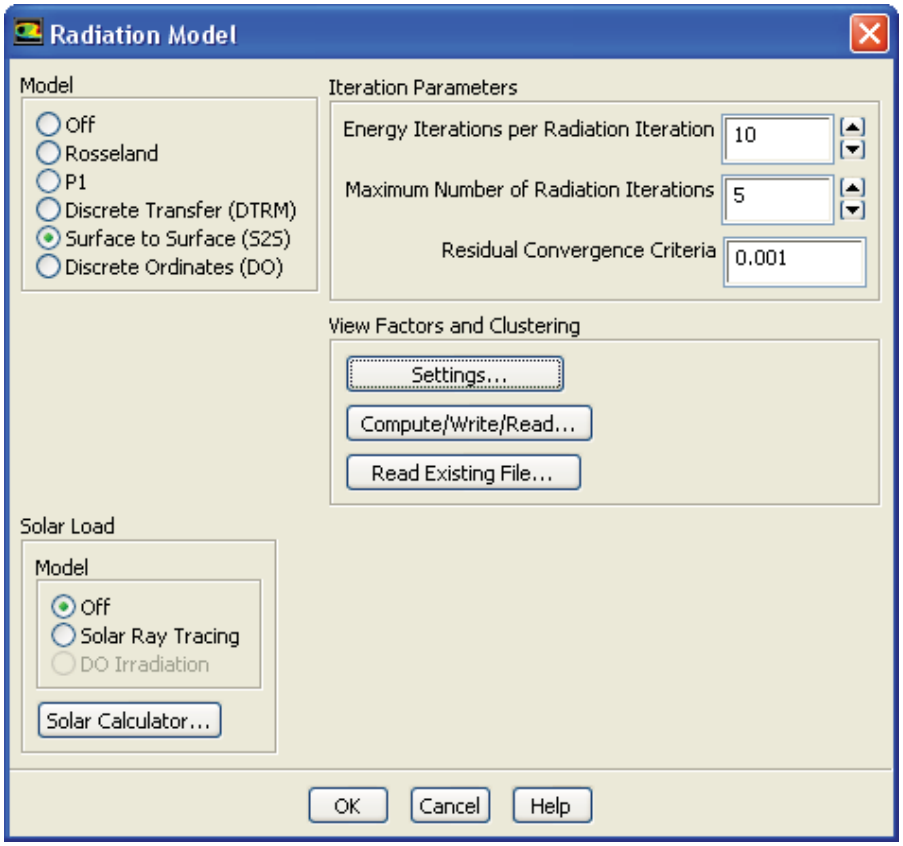


Step 3: Models

1. Enable the energy equation.



2. Set up the Surface to Surface (S2S) radiation model.



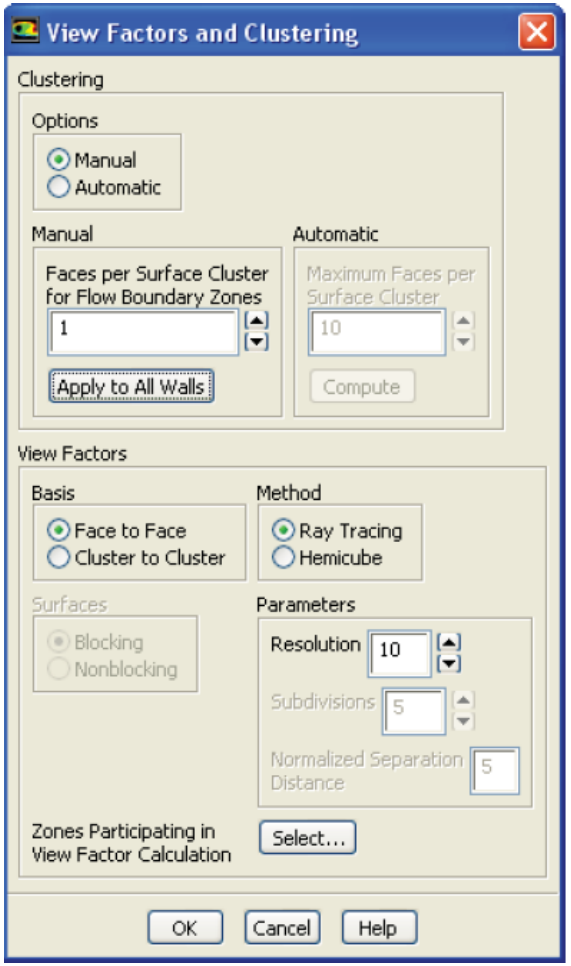
Step 3: Model details

The surface-to-surface (S2S) radiation model can be used to account for the radiation exchange in an enclosure of gray-diffuse surfaces. The energy exchange between two surfaces depends in part on their size, separation distance, and orientation. These parameters are accounted for by the geometric function called "view factor".

The S2S model assumes that all surfaces are gray and diffuse. Thus according to the gray-body model, if a certain amount of radiation is incident on a surface, then a fraction is reflected, a fraction is absorbed, and a fraction is transmitted. The main assumption of the S2S model is that any absorption, emission, or scattering of radiation by the medium can be ignored. Therefore only "surface-to surface" radiation is considered for analysis.

Step 3: Models

Click the Settings... button to open the View Factors and Clustering dialog box. You will define the view factor and cluster parameters.



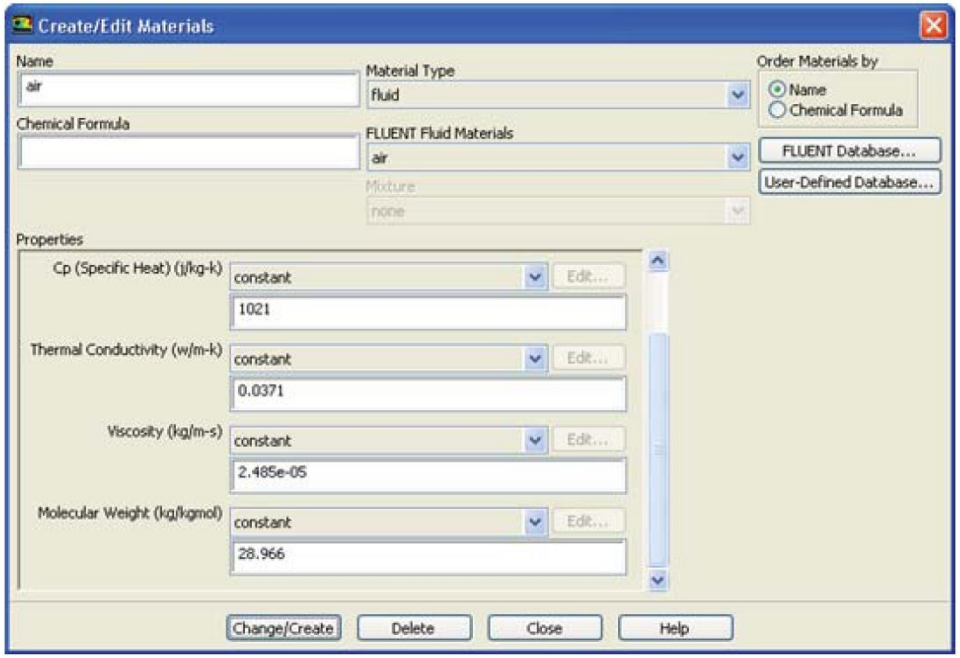
The S2S radiation model is computationally very expensive when there are a large number of radiating surfaces. The number of radiating surfaces is reduced by clustering surfaces into surface "clusters". The surface clusters are made by starting from a face and adding its neighbors and their neighbors until a specified number of faces per surface cluster is collected.

For a small problem, the default value of 1 for Faces per Surface Cluster for Flow Boundary Zones is acceptable. For a large problem you can increase this number to reduce the memory requirement for the view factor file that is saved in a later step. This may also lead to some reduction in the computational expense. However, this is at the cost of some accuracy.

Step 4: Materials

Gas properties

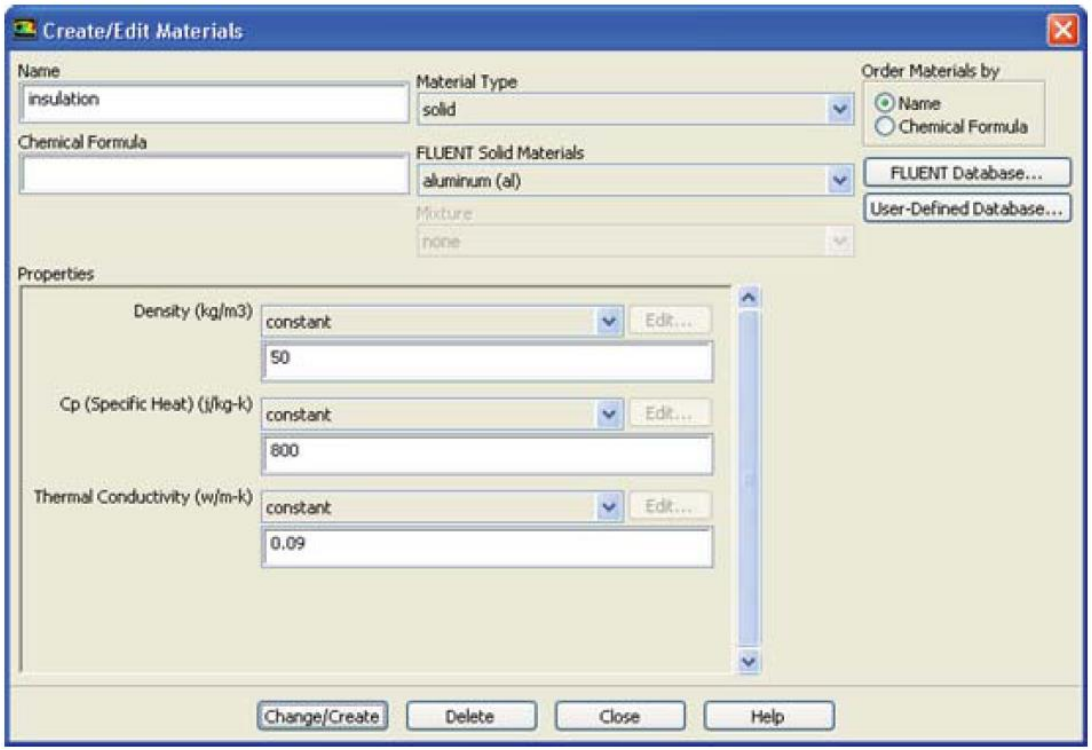
a. Select incompressible-ideal-gas from the Density drop-down list.
b. Enter 1021 J/kgK for Cp (Specific Heat).
c. Enter 0.0371 W/mK for Thermal Conductivity.
d. Enter 2.485e-05 kg/ms for Viscosity.
e. Retain the default value of 28.966 kg/kmol for Molecular Weight.
f. Click Change/Create and close the Create/Edit Materials dialog box.



Step 4: Materials

Solid properties

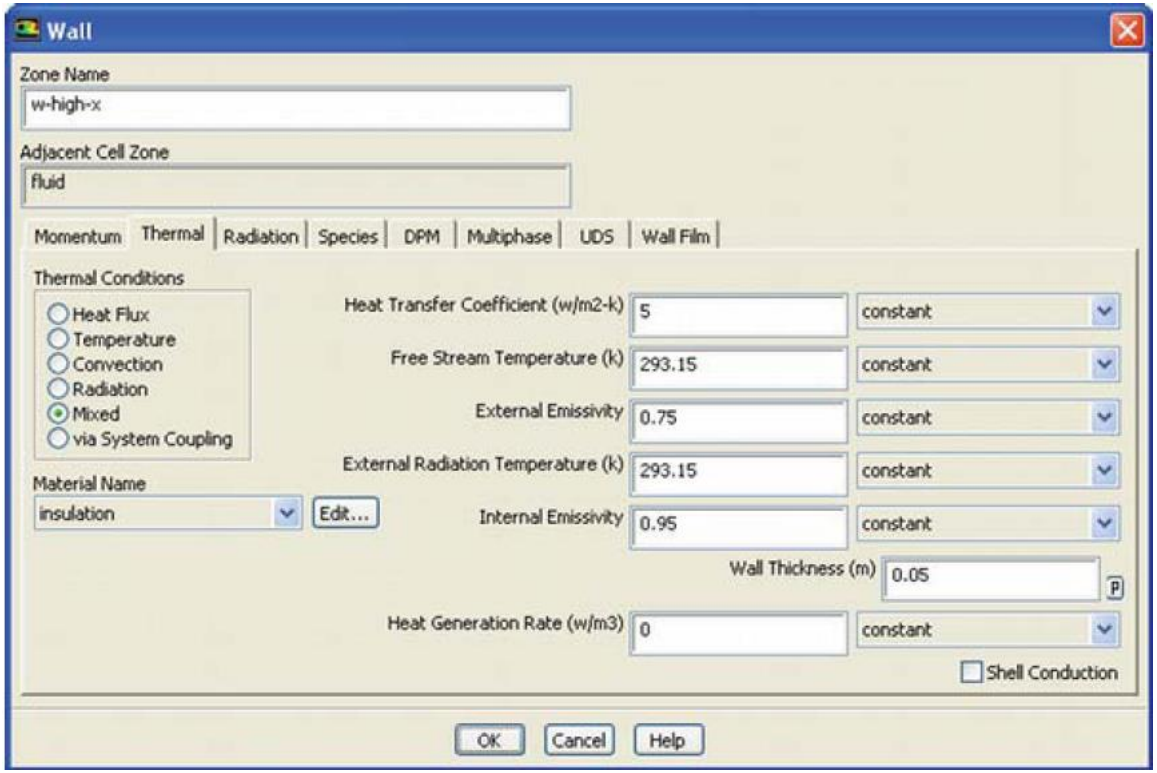
Materials → Solid → Create/Edit...

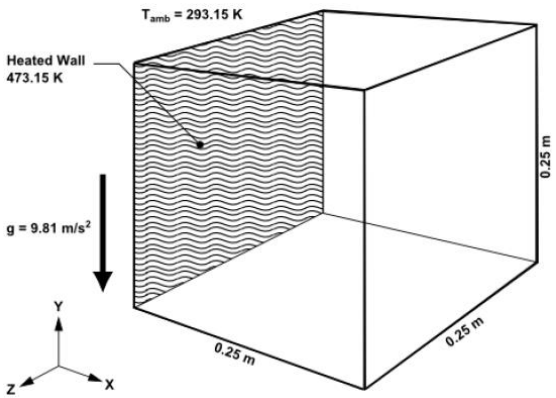


a. Enter insulation for **Name**.
b. Delete the entry in the **Chemical Formula** field.
c. Enter 50 kg/m^3 for **Density**.
d. Enter 800 J/kgK for **Cp (Specific Heat)**.
e. Enter 0.09 W/mK for **Thermal Conductivity**.

Step 5: Boundary Conditions

Set the boundary conditions for the front wall (**w-high-x**).

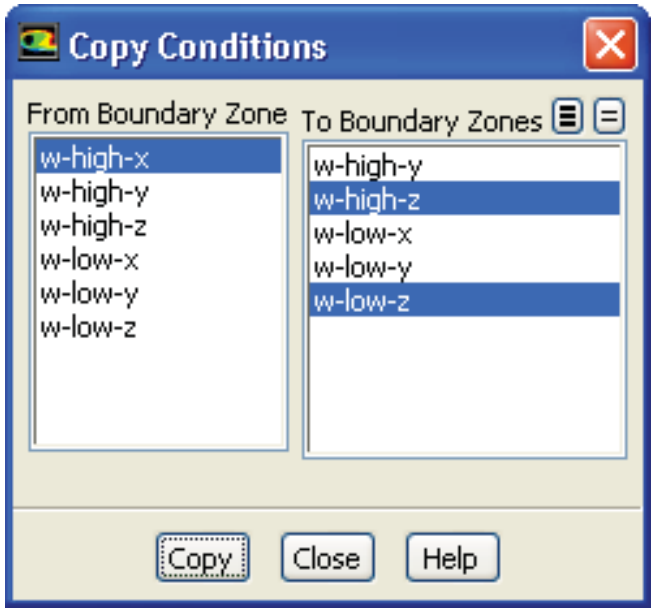


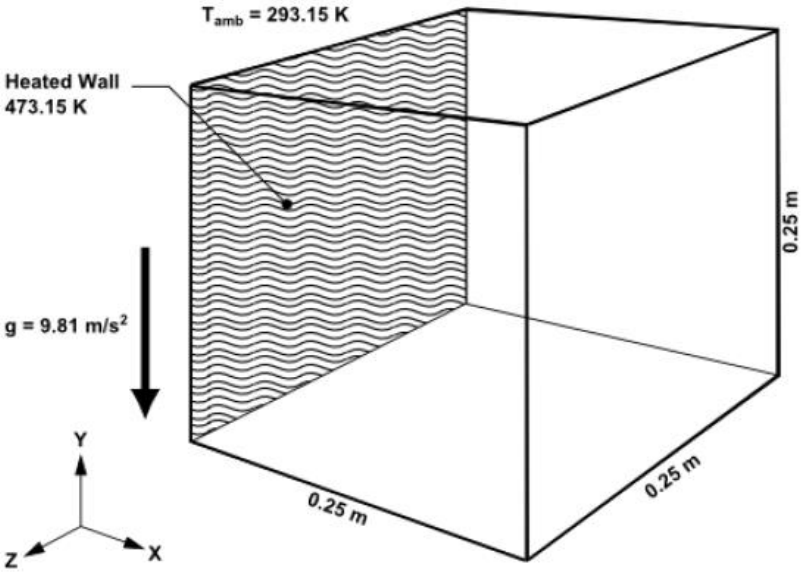


Step 5: Boundary Conditions

Copy boundary conditions to define the side walls **w-high-z** and **w-low-z**.

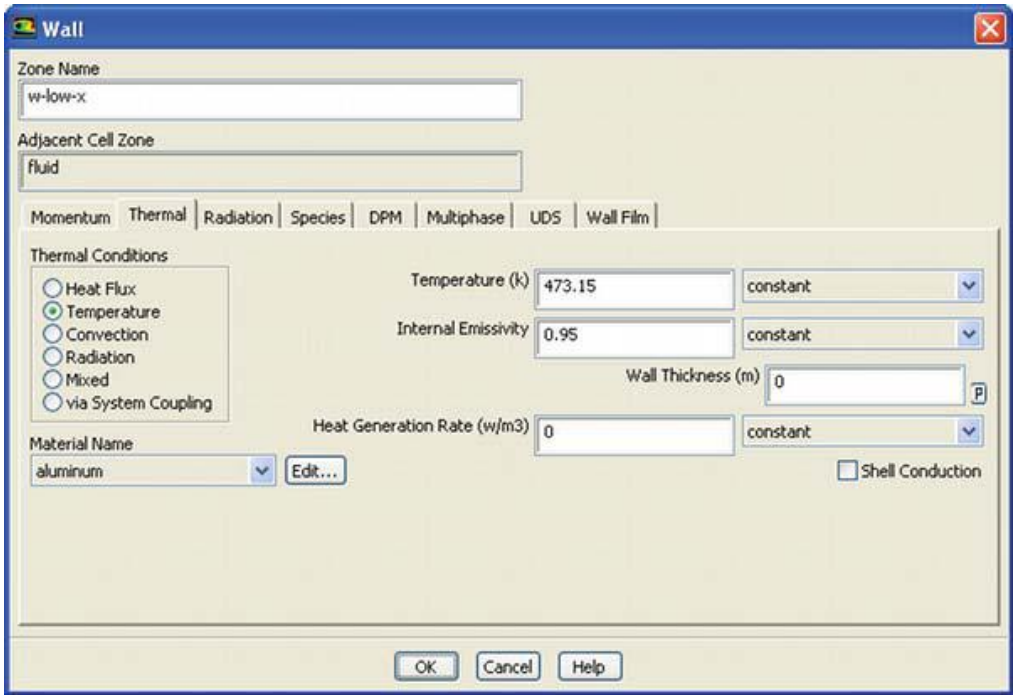
Boundary Conditions → Copy...





Step 5: Boundary Conditions

Set the boundary conditions for the front wall (**w-low-x**).



a. Click the Thermal tab and select Temperature from the Thermal Conditions list.

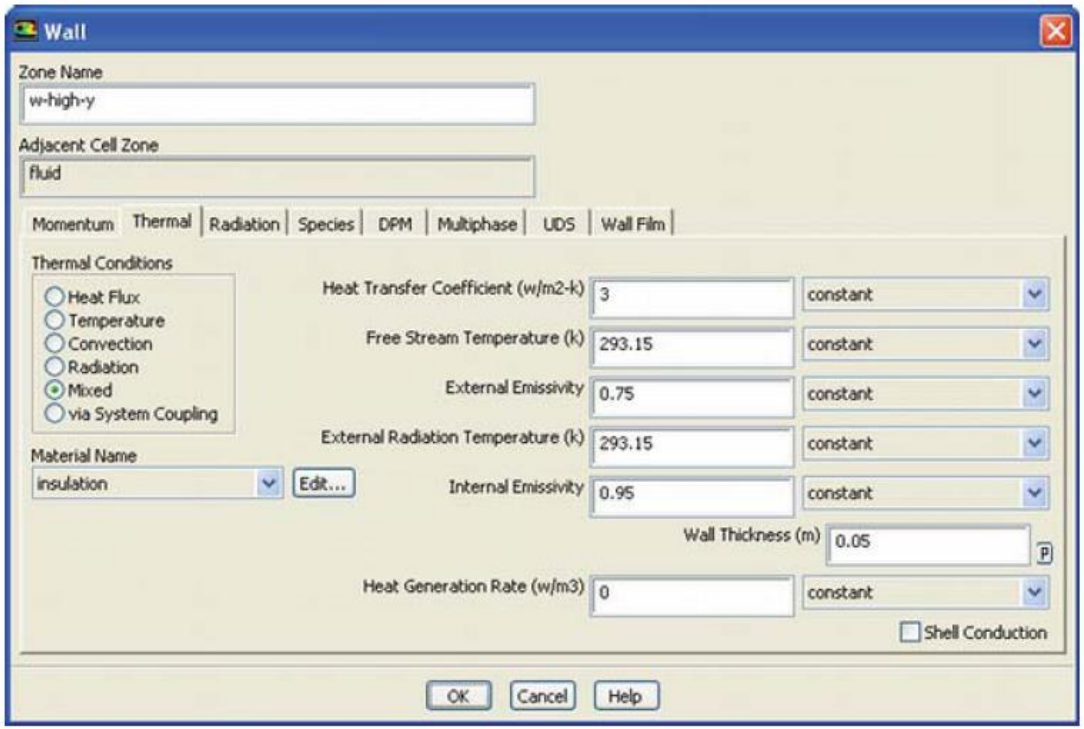
b. Retain the default selection of aluminum from the Material Name drop-down list.

c. Enter 473.15 K for Temperature.

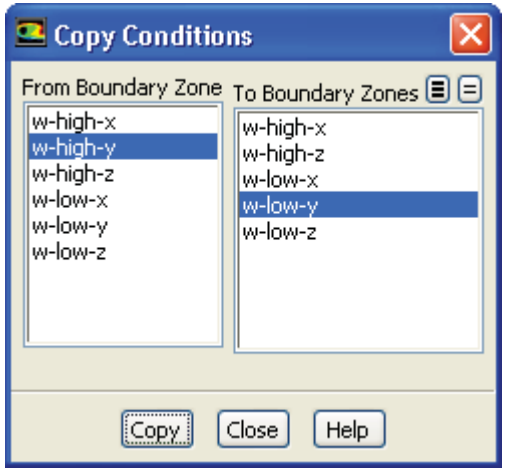
d. Enter 0.95 for Internal Emissivity.

Step 5: Boundary Conditions

Set the boundary conditions for the top wall (**w-high-y**).

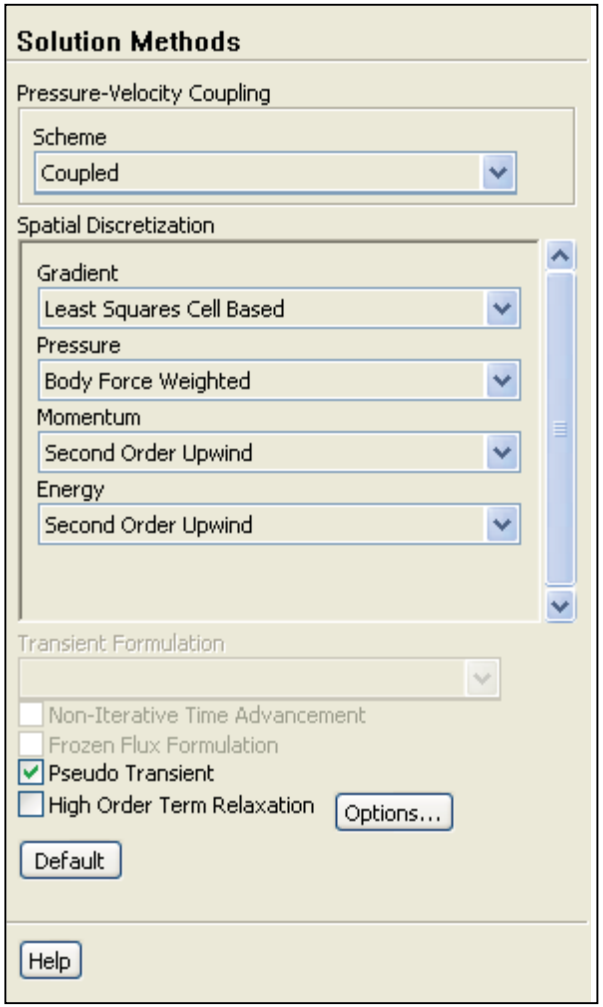


Copy boundary conditions to define the bottom wall (**w-low-y**).



Step 6: Solution

1. Set the solution parameters.



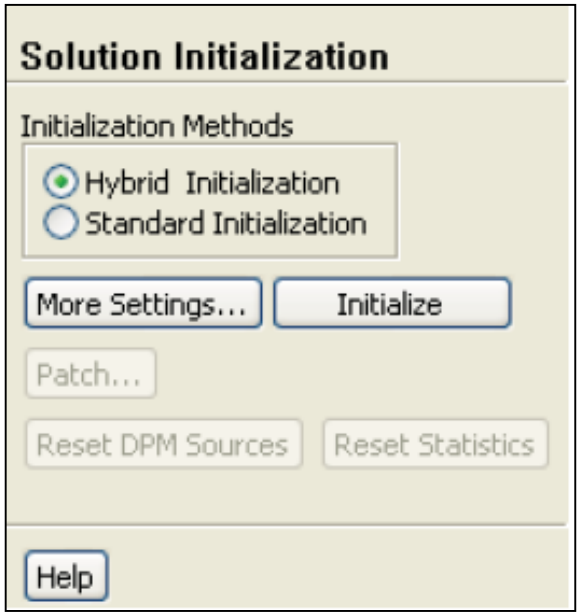
a. Select Coupled from the Scheme drop-down list in the Pressure-Velocity Coupling group box.

b. Select Body Force Weighted from the Pressure drop-down list in the Spatial Discretization group box.

c. Retain the default selection of Second Order Upwind from the Momentum and Energy drop-down lists.

Step 6: Solution

2. Initialize the solution.



a. Retain the default selection of **Hybrid Initialization** from the **Initialization Methods** list.

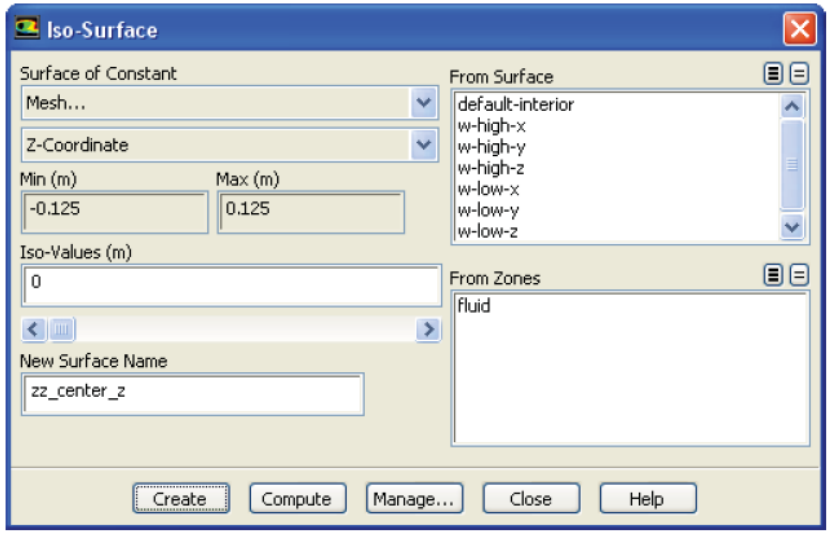
b. Click **Initialize**.

Step 6: Solution

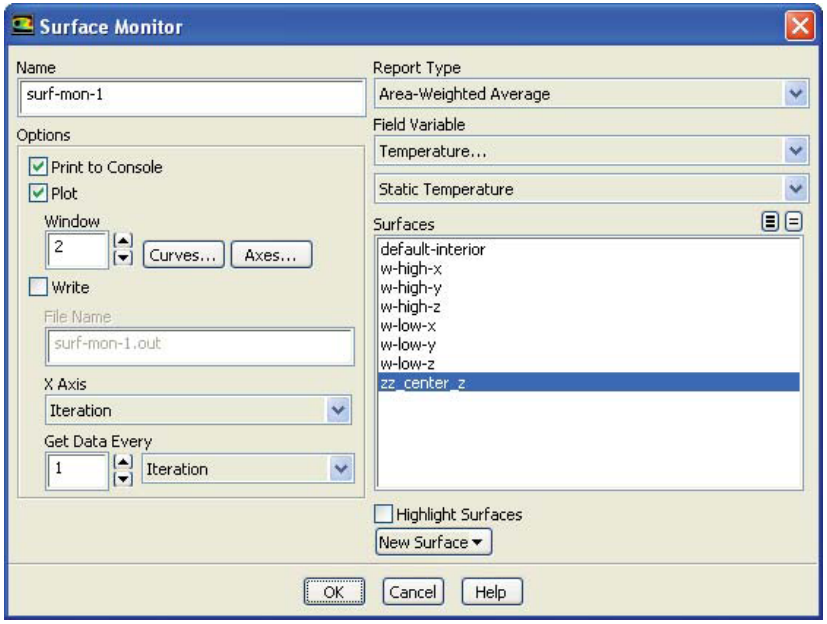
3. Define a surface monitor to aid in judging convergence.

Define a surface monitor to aid in judging convergence. *It is good practice to use monitors of physical solution quantities together with residual monitors when determining whether a solution is converged. In this step you will set up a surface monitor of the average temperature on the z=0 plane.*

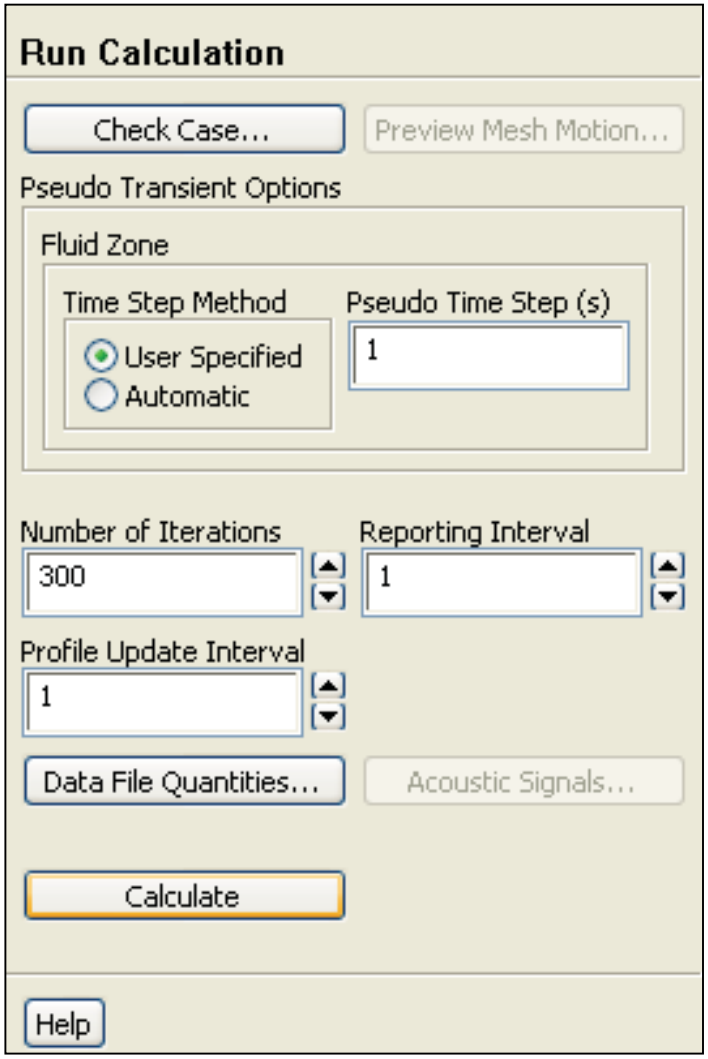
a. Creating a surface



b. Create the surface monitor.



Step 6: Solution



a. Select **User Specified** from the **Time Step Method** list.

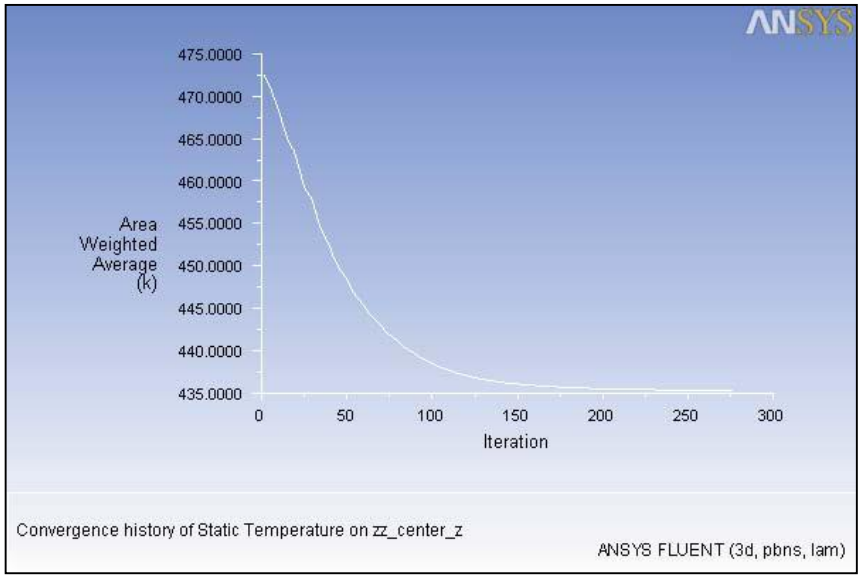
b. Retain the default value of 1 for **Pseudo Time Step**.

c. Enter 300 for **Number of Iterations**.

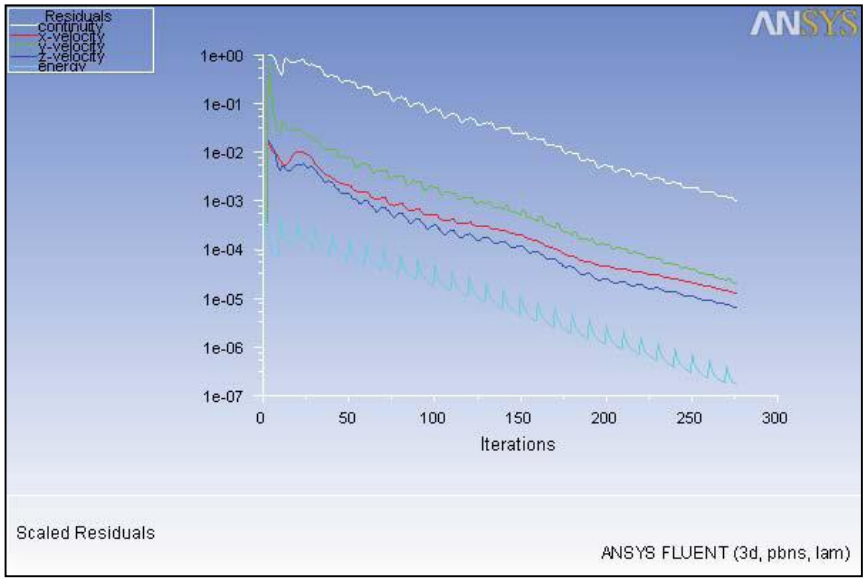
d. Click **Calculate**.

Step 6: Solution

Temperature Surface Monitor



Scaled Residuals

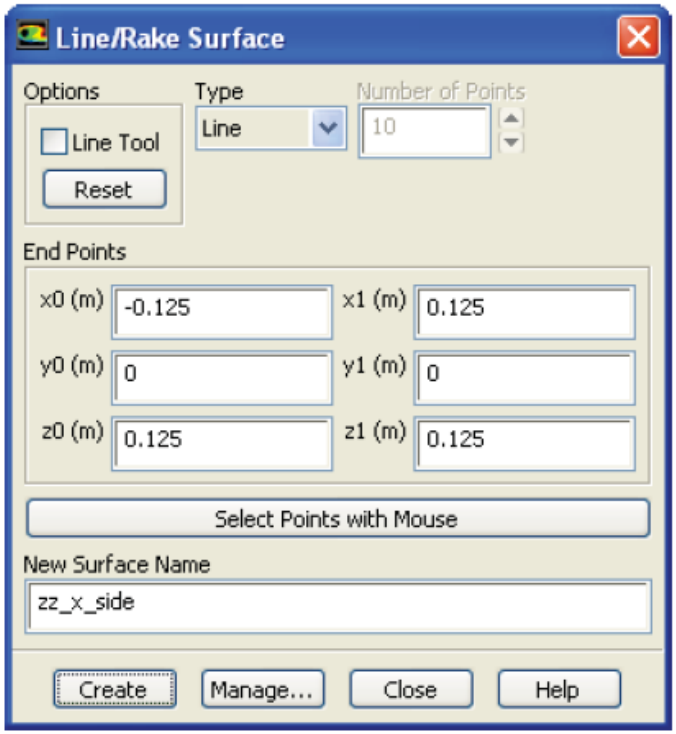


The surface monitor history shows that the average temperature on zz_center_z has stabilized, thus confirming that the solution has indeed reached convergence. You can view the behavior of the residuals by selecting Scaled Residuals from the graphics window drop-down list.

Step 7: Postprocessing

1. Create a new surface, **zz_x_side**, which will be used later to plot wall temperature.

Surface → Line/Rake...



a. Enter (-0.125, 0, 0.125) for (x0, y0, z0), respectively.

b. Enter (0.125, 0, 0.125) for (x1, y1, z1), respectively.

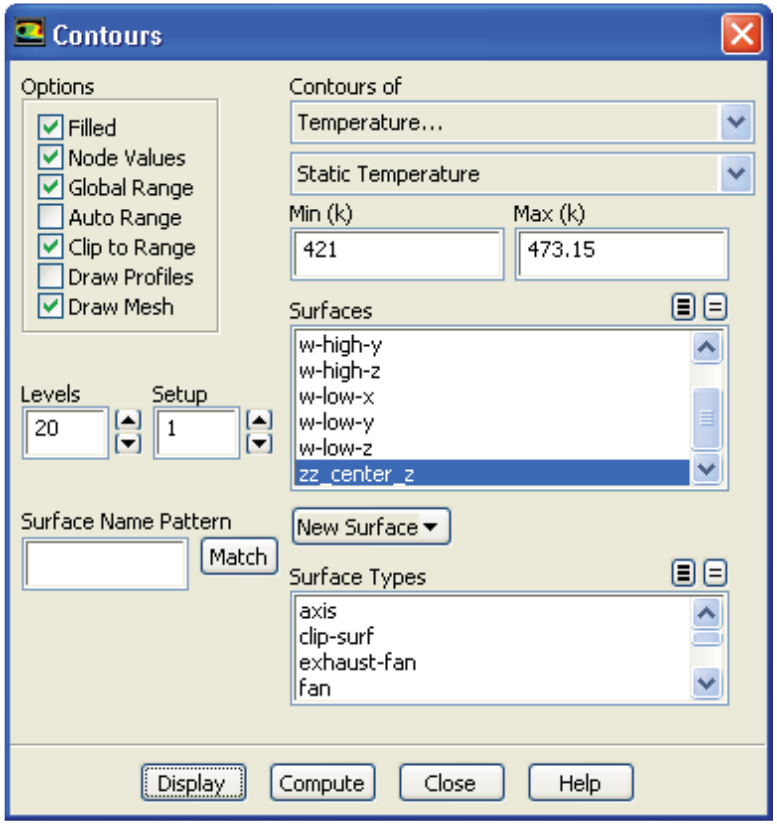
c. Enter zz_x_side for New Surface Name.

d. Click Create and close the Line/Rake Surface dialog box.

Step 7: Postprocessing

2. Display contours of static temperature.

Graphics and Animations → Contours → Set Up...



a. Enable the **Filled** option in the **Options** group box.

b. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

c. Select **zz_center_z** from the **Surfaces** selection list.

d. Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.

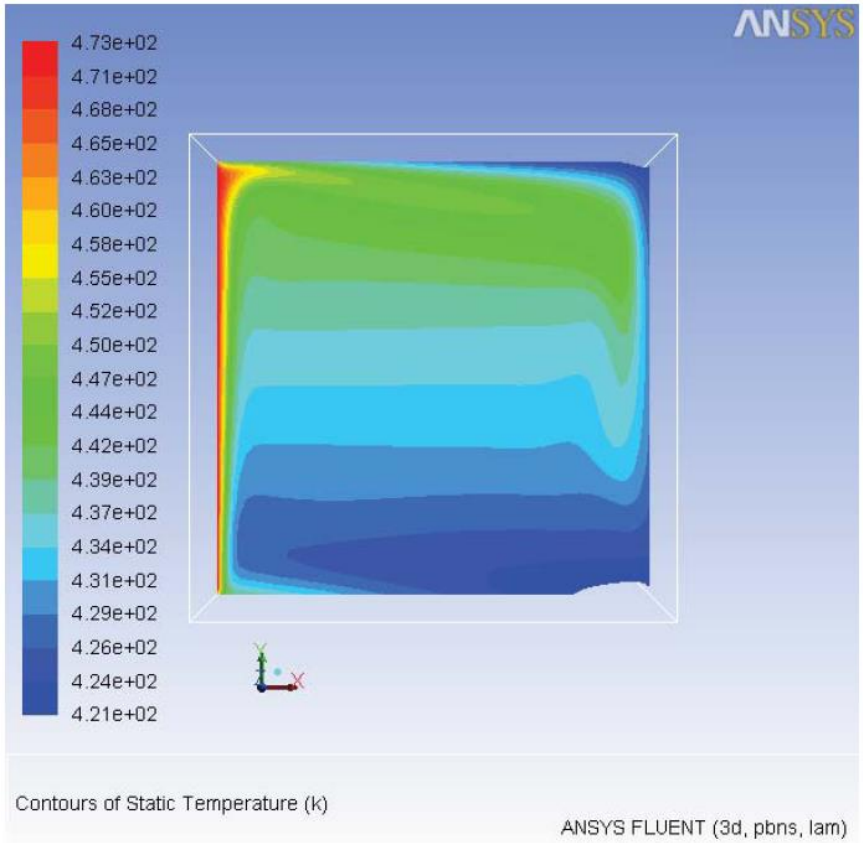
i. Select **Outline** from the **Edge Type** list.

ii. Click **Display** and close the **Mesh Display** dialog box.

e. Disable the **Auto Range** option.

f. Enter 421 K for **Min** and 473.15 K for **Max**.

Step 7: Postprocessing

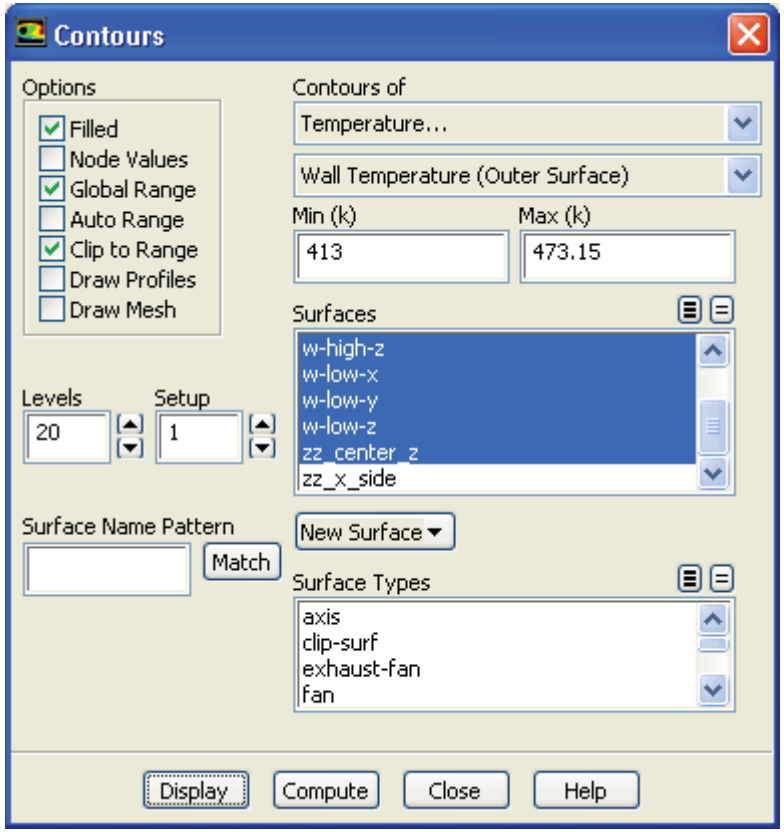


A regular check for most buoyant cases is to look for evidence of stratification in the temperature field. This is observed as nearly horizontal bands of similar temperature. These may be broken or disturbed by buoyant plumes. For this case you can expect reasonable stratification with some disturbance at the vertical walls where the air is driven around.

Step 7: Postprocessing

3. Display contours of wall temperature (outer surface).

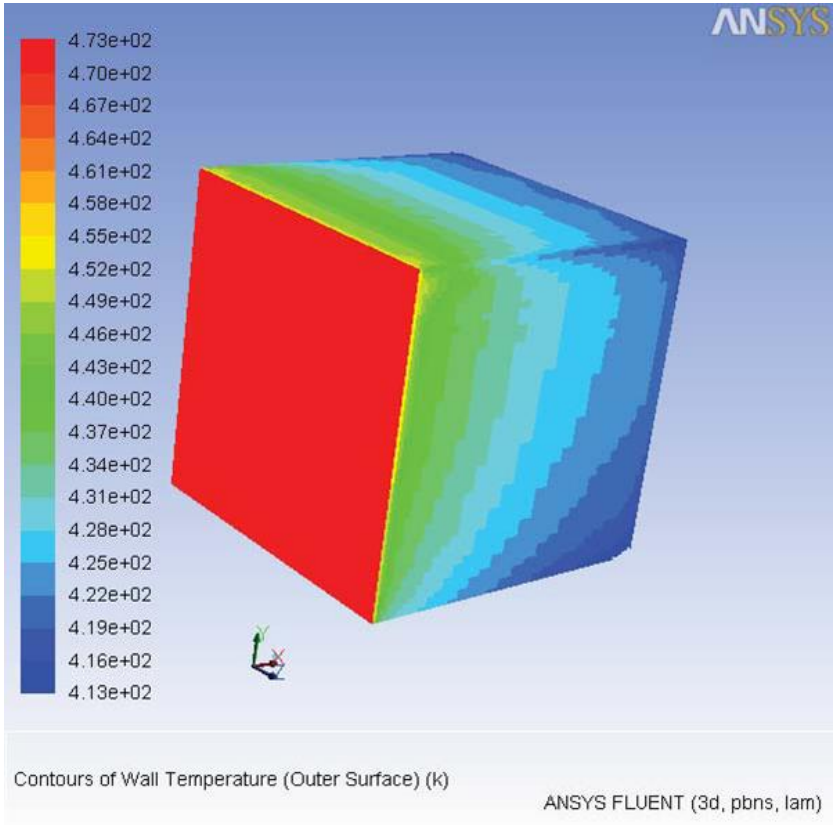
Graphics and Animations → Contours → Set Up...



a. Ensure that the Filled option is enabled in the Options group box.
b. Disable the Node Values option.
c. Select Temperature... and Wall Temperature (Outer Surface) from the Contours of drop-downlists.
d. Select all surfaces except default-interior and zz_x_side in the Surfaces selection list.
e. Disable the Auto Range and Draw Mesh options.
f. Enter 413 K for Min and 473.15 K for Max.

Step 7: Postprocessing

Contours of Wall Temperature

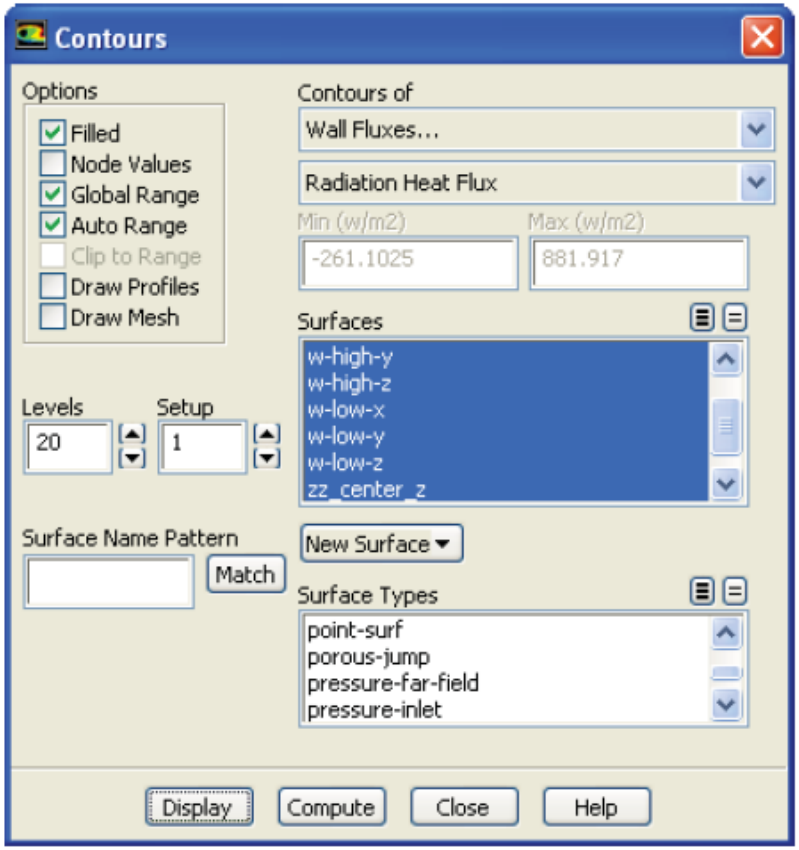


In the context of the Wall Temperature field variables, Outer Surface refers to whichever surface of the wall is in contact with the fluid domain. This may or may not correspond to an “outer” wall of the problem geometry.

Step 7: Postprocessing

4. Display contours of radiation heat flux.

Graphics and Animations → Contours → Set Up...



a. Ensure that the **Filled** option is enabled in the **Options** group box.

b. Select **Wall Fluxes...** and **Radiation Heat Flux** from the **Contours of** drop-down list.

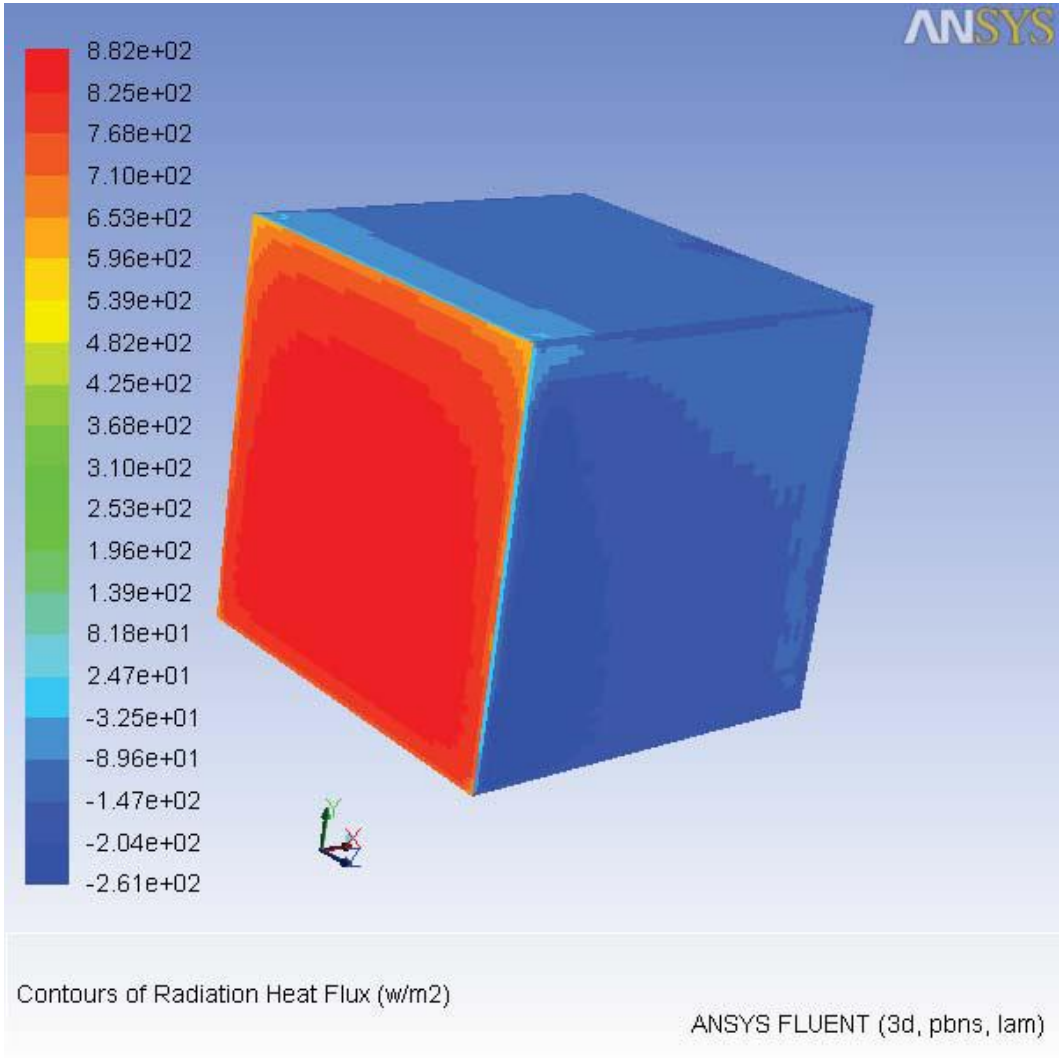
c. Make sure that all surfaces except **default-interior** and **zz_x_side** are selected in the **Surfaces** selection list.

d. Click **Display**.

e. Close the **Contours** dialog box.

Step 7: Postprocessing

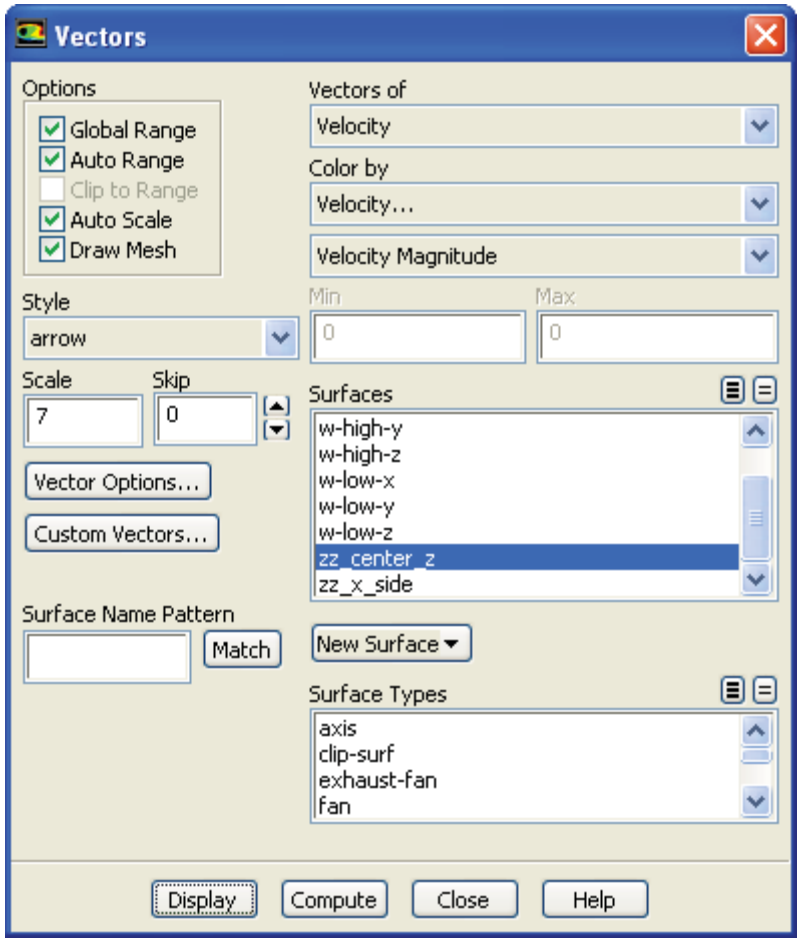
Contours of Radiation Heat Flux

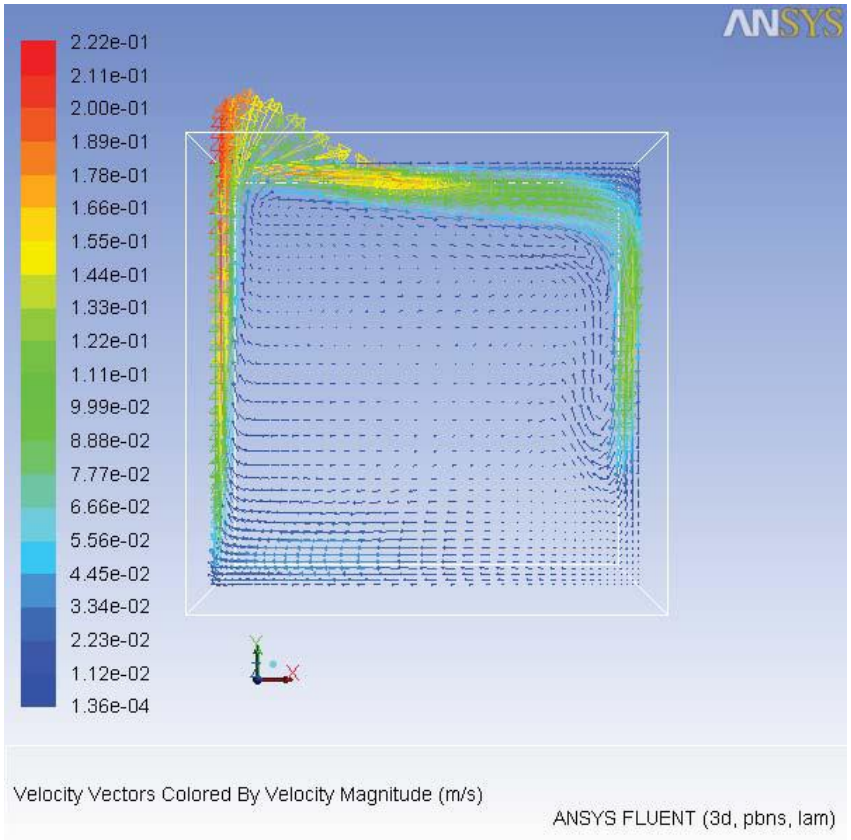


Step 7: Postprocessing

5. Display vectors of velocity magnitude.

Graphics and Animations → Vectors → Set Up...

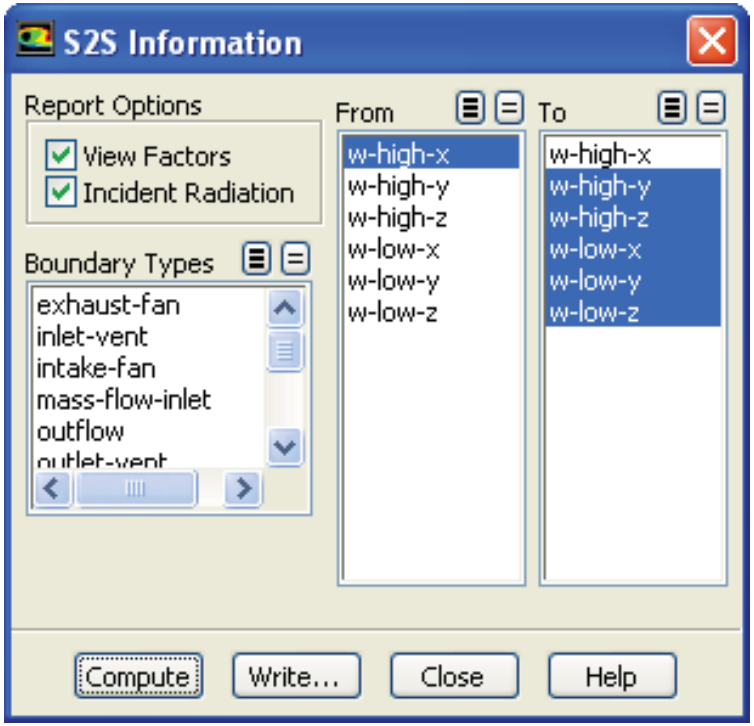




Step 7: Postprocessing

6. Compute view factors and radiation emitted from the front wall (w-high-x) to all other walls.

Report → S2S Information...



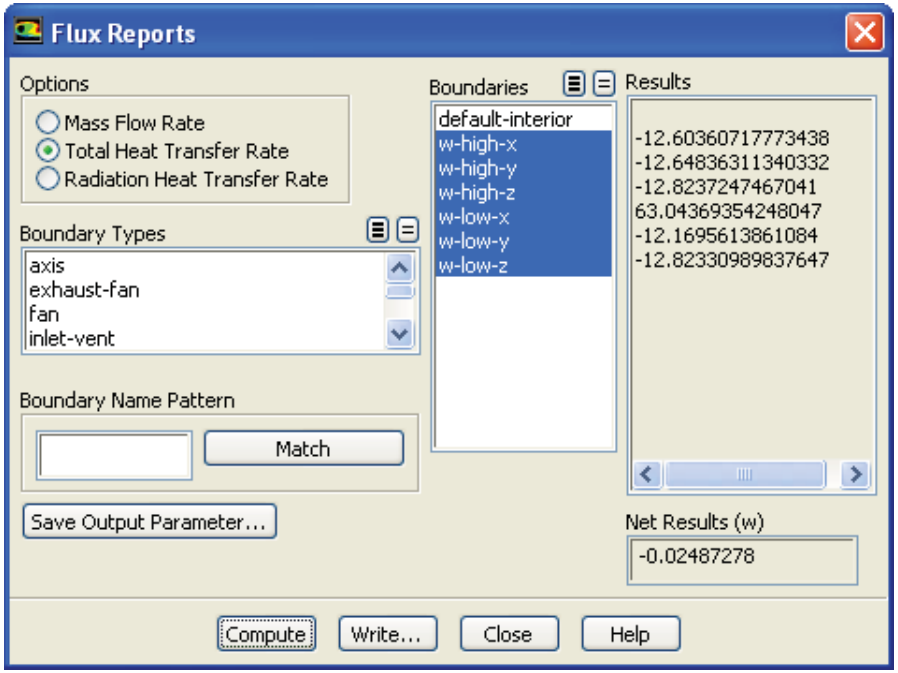
a. Ensure that the View Factors option is enabled in the Report Options group box.
b. Enable the Incident Radiation option.
c. Select w-high-x from the From selection list.
d. Select all zones except w-high-x from the To selection list.
e. Click Compute and close the S2S Information dialog box.

Step 7: Postprocessing

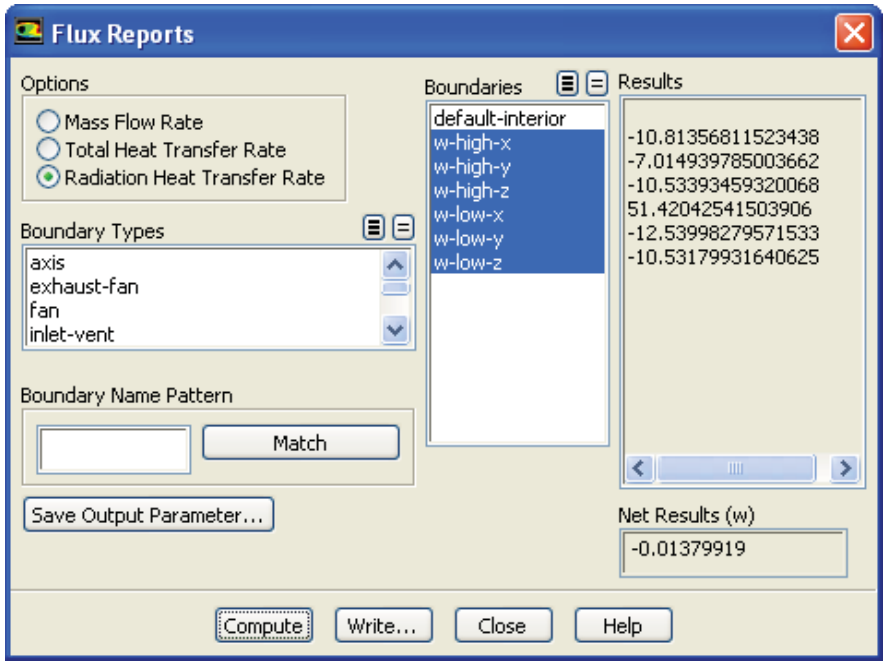
7. Compute the heat transfer rate.

Reports → Fluxes → Set Up...

Total heat transfer rate



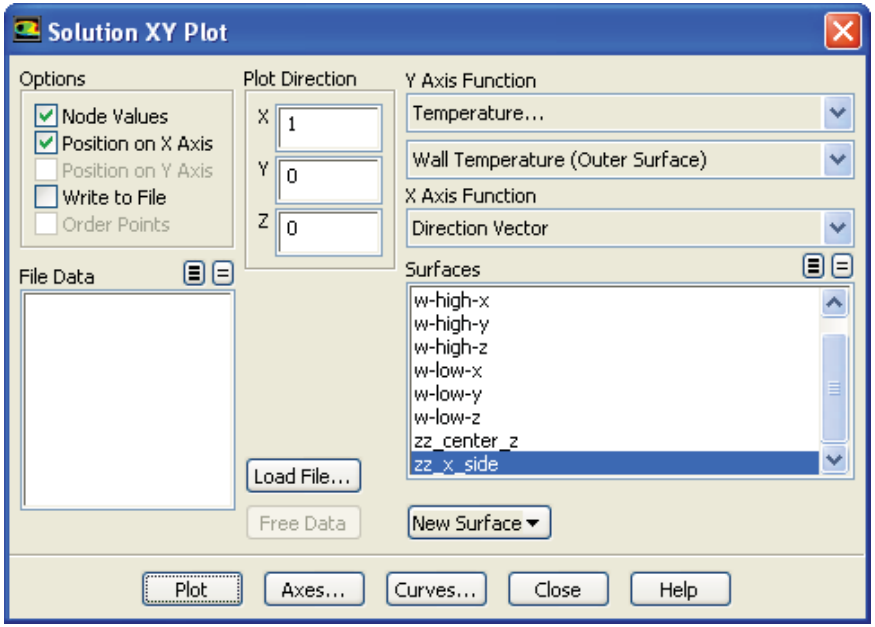
Radiation heat transfer rate

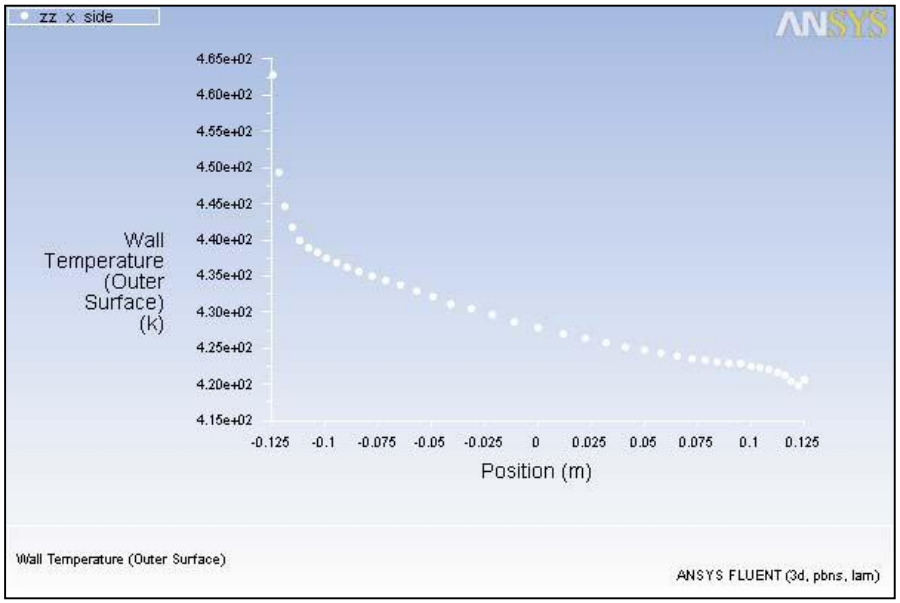


Step 7: Postprocessing

8. Display the temperature profile for the side wall.

Plots → XY Plot → Set Up...





Temperature Profile Along Outer Surface

Step 8: Changing radiation settings

Step 8: Compare the Contour Plots after Varying Radiating Surfaces

1. Increase the number of faces per cluster to 10.

Models → Radiation → Edit...

a. Click the Settings... button to open the View Factors and Clustering dialog box.

b. Click the Compute/Write/Read... button to open the Select File dialog box and to compute the view factors.

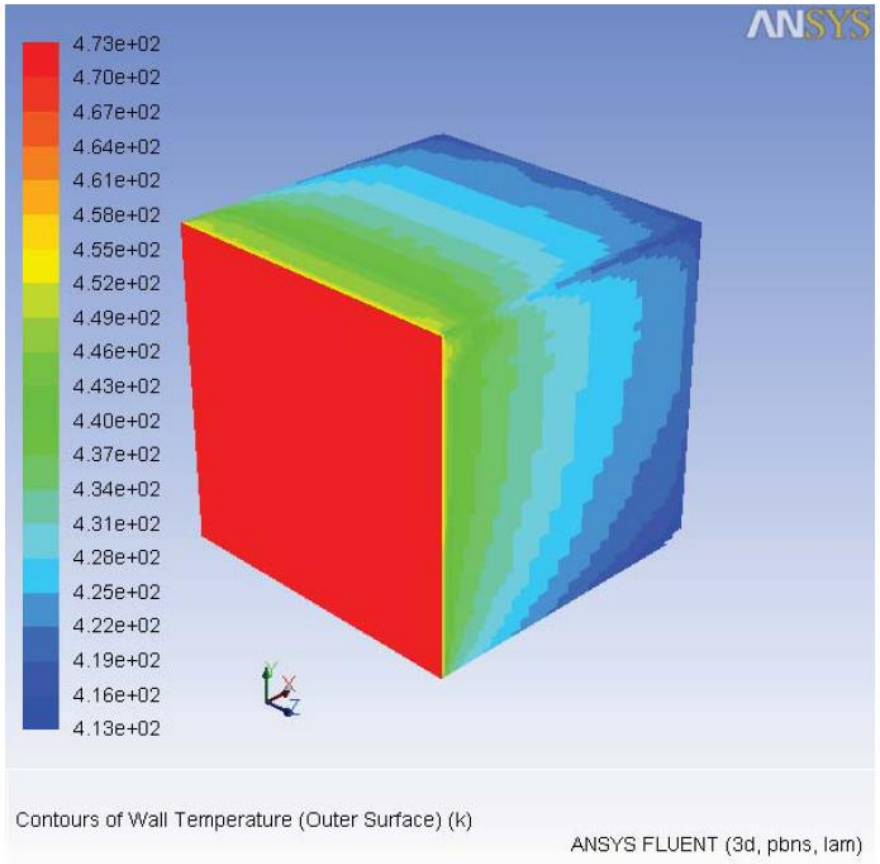
2. Initialize the solution.

3. Start the calculation by requesting 300 iterations.

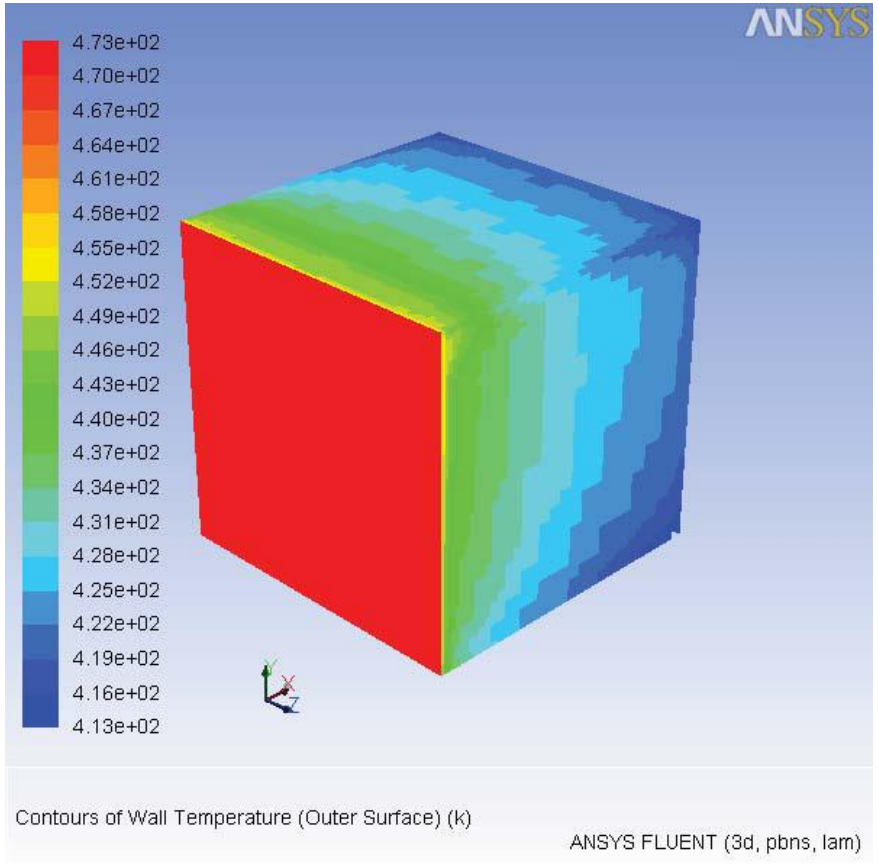
Repeat the procedure outlined in Step 8: for 100, 400, 800, and 1600 faces per surface cluster.

Step 8: Changing radiation settings

Contours of Wall Temperature (Outer Surface): 1 Face per Surface Cluster

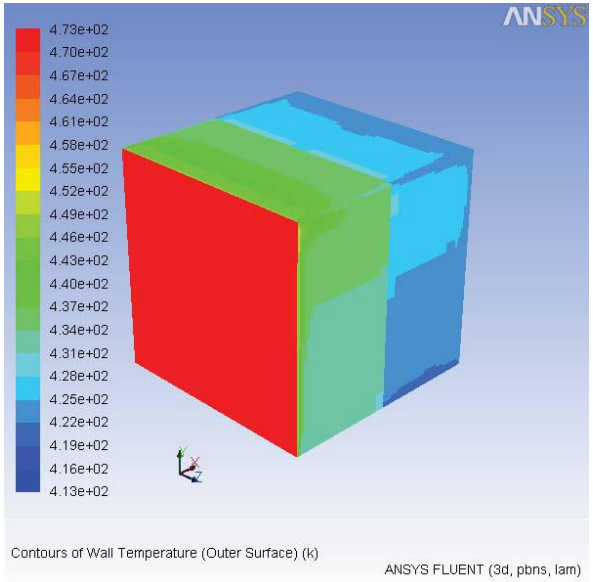


Contours of Wall Temperature (Outer Surface): 100 Face per Surface Cluster

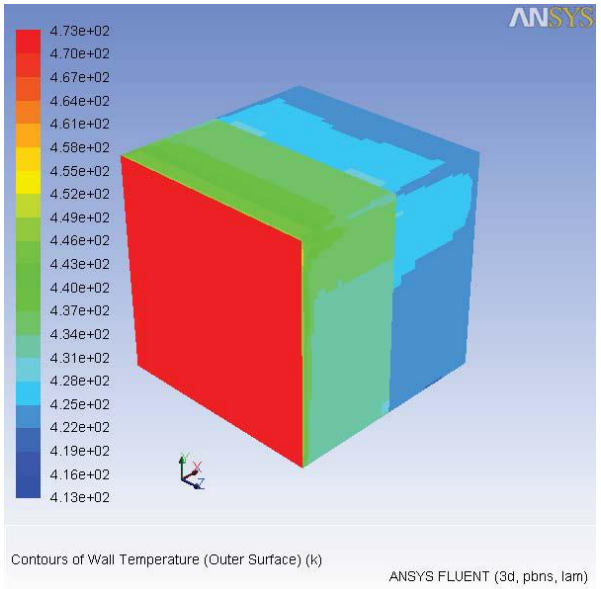


Step 8: Changing radiation settings

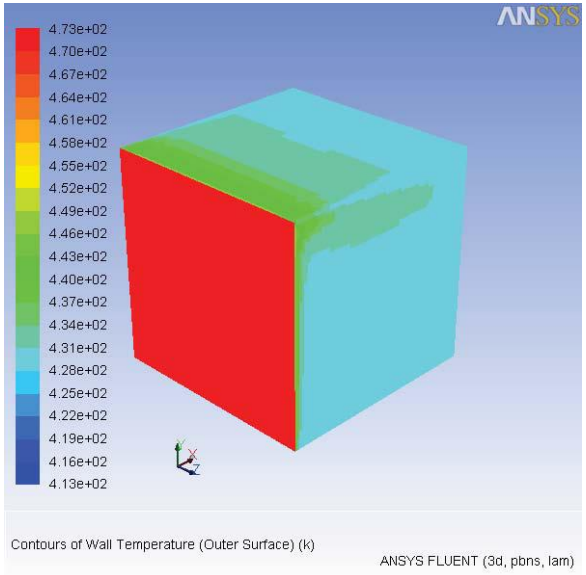
Contours of Wall Temperature (Outer Surface): 400 Face per Surface Cluster



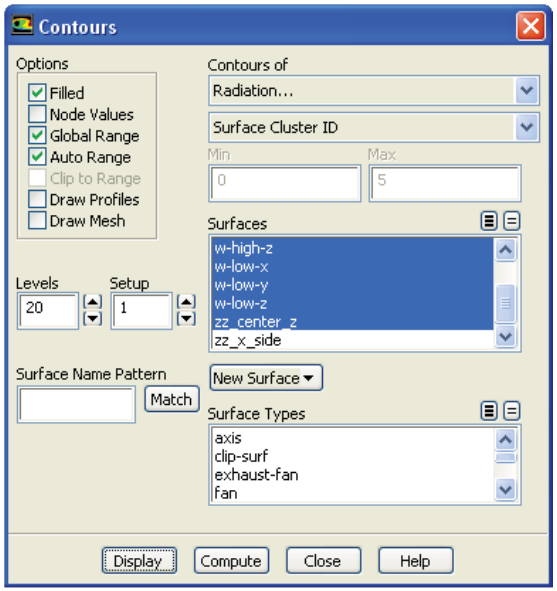
Contours of Wall Temperature (Outer Surface): 800 Face per Surface Cluster



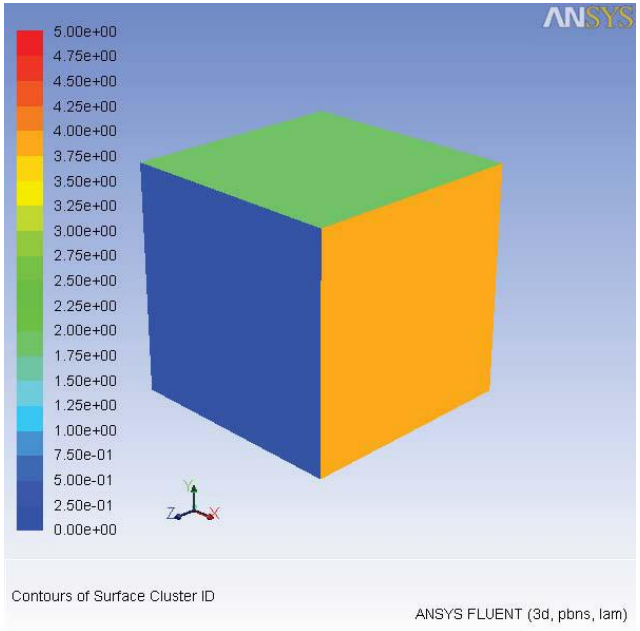
Contours of Wall Temperature (Outer Surface): 1600 Face per Surface Cluster



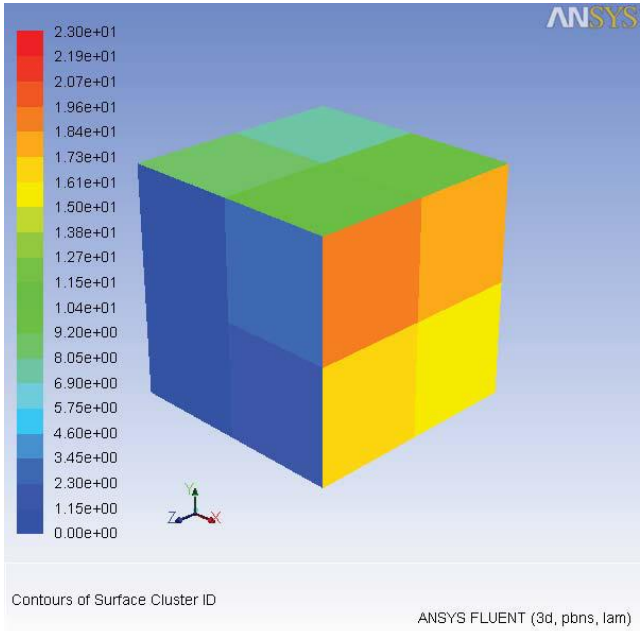
Step 8: Changing radiation settings



Contours of Surface Cluster ID—1600 Faces per Surface Cluster (FPSC)



Contours of Surface Cluster ID—400 FPSC



Step 9: Partial enclosure settings

As mentioned previously, when the S2S model is used, one also has the option to define a "partial enclosure"; that means, one can disable the view factor calculation for walls with negligible emission/absorption, or walls that have a uniform temperature.

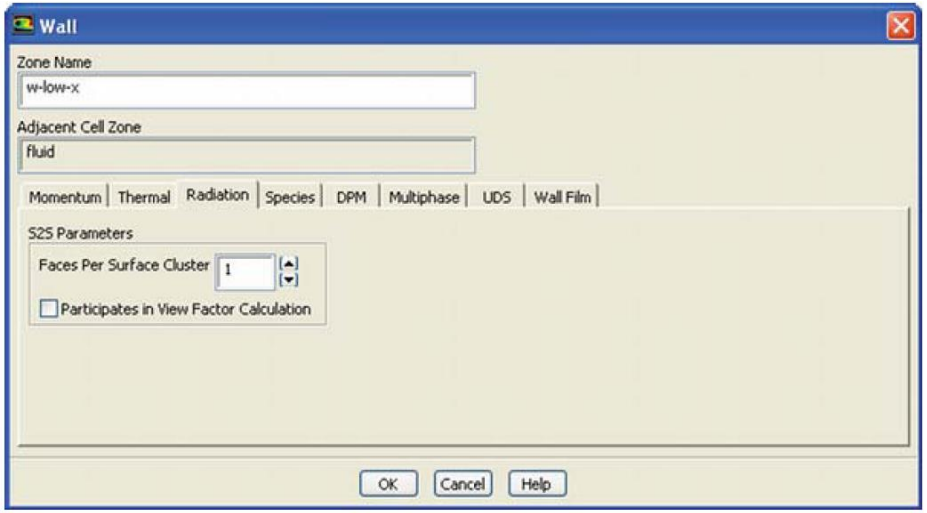
Even though the view factor will not be computed for these walls, they will still emit radiation at a fixed temperature called the "partial enclosure temperature". The main advantage of this is to speed up the view factor calculation.

In the steps that follow, one has to specify the radiating wall (w-low-x) as a boundary zone that is not participating in the S2S radiation model. Consequently, one has to specify the partial enclosure temperature for the wall.

Step 9: Partial enclosure settings

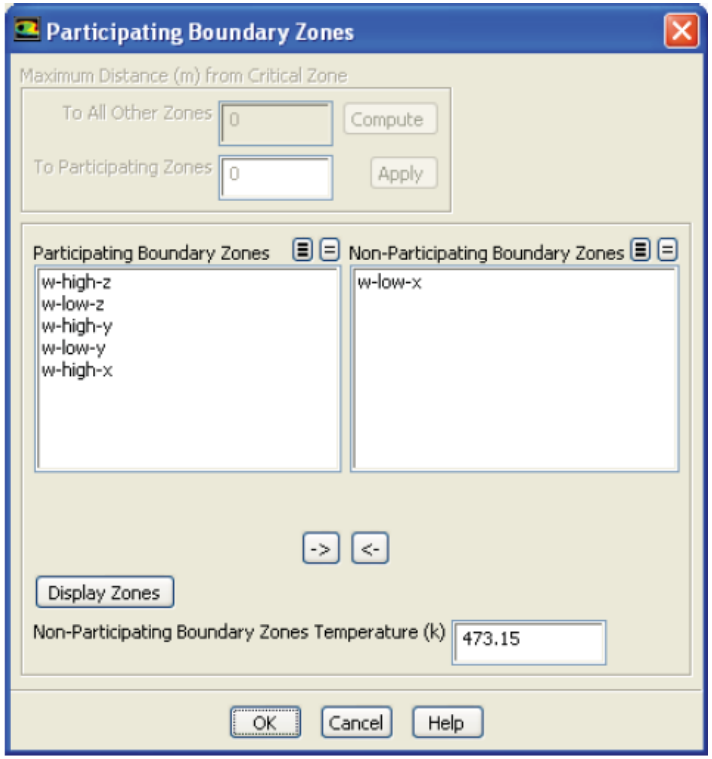
Boundary Conditions → w-low-x → Edit...

a. Click the Radiation tab.
b. Disable the Participates in View Factor Calculation option in the S2S Parameters group box.
c. Click OK to close the Wall dialog box.



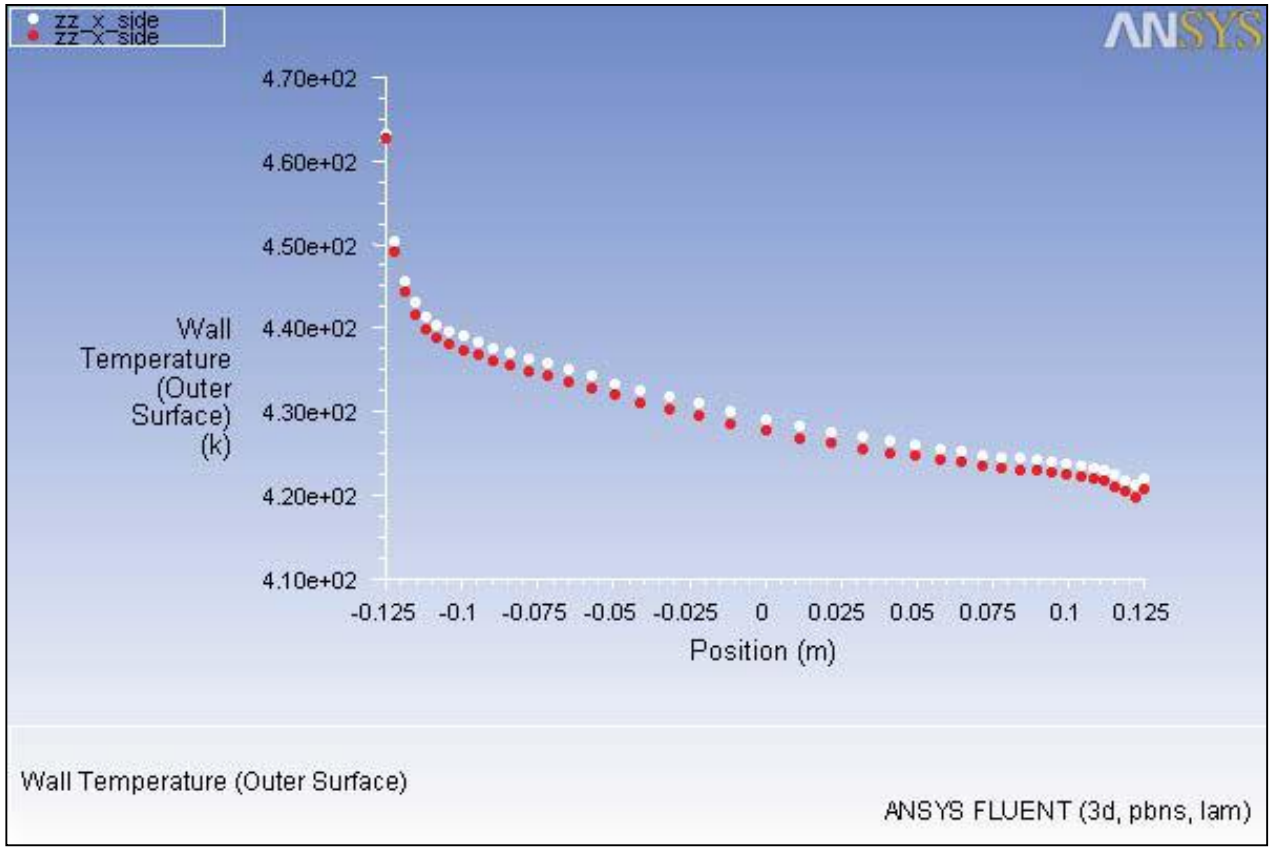
Models → Radiation → Edit...

a. Click the **Settings...** button to open the **View Factors and Clustering** dialog box.
b. Click the **Select...** button to open the **Participating Boundary Zones** dialog box.



Step 9: Partial enclosure settings

Temperature Profile Comparison on Outer Surface



Temperature Profile Comparison on Outer Surface further confirms that the use of a partial enclosure did not significantly affect the results.

Summary

In this example, we combined natural convection and radiation in a three-dimensional square box and compared how varying the settings of the surface-to-surface (S2S) radiation model affected the results. The S2S radiation model is appropriate for modeling the enclosure radiative transfer without participating media, whereas the methods for participating radiation may not always be efficient.

Hopefully we are ready for the computer session!