

# Surface to Surface Radiation Benchmarks

Randy van Eck, Michiel Klep and Jos van Schijndel\*

Eindhoven University of Technology

\*Corresponding author: P.O.Box 513 Eindhoven, Netherlands, A.W.M.Schijndel@tue.nl

**Abstract:** The paper presents a student guide on how to implement surface to surface radiation within COMSOL Multiphysics 5.2 for case studies found within the built environment. The paper is based on the work of Eck and Klep (2016). We included four benchmarks: (1) Radiation in a triangular cavity with infinite length; (2) Radiation between two infinitely long rectangular plates; (3) Radiation in a three dimensional rectangular enclosure; (4) Radiation in an ice rink building. We conclude that Comsol benchmark results are satisfactory. All files are available at the HAMLab (2016) website.

**Keywords:** Surface, radiation, heat transfer, benchmark.

## 1. Introduction

COMSOL computes the total radiative flux leaving a surface (radiosity). This approach is valid only for isothermal surfaces and objects. This means that the radiative properties (emissivity  $[\epsilon]$ , absorptivity  $[\alpha]$  reflectivity  $[\rho]$ ) remain constant. This applies to cases where  $\alpha = \epsilon$ . These conditions are satisfied for: i) cases in which the irradiation is diffuse, ii) the surface is diffuse, iii) the radiation spectrum only depends on the temperature of the object (black-body radiation) or iv) if the radiation properties are independent of the wavelength. Overall, the radiosity can be defined as:

$$Q_{ij} = A_i F_{ij} (J_i - J_j) \quad (1)$$

Where:

$Q_{ij}$  = Power transmitted from body i to body j [W]

$A_i$  = the surface area of body i [m<sup>2</sup>]

$F_{ij}$  = the view factor from body i to body j [-]

$J_i$  = total radiative flux leaving surface i [W/m<sup>2</sup>]

$J_j$  = total radiative flux leaving surface j [W/m<sup>2</sup>]

For a certain temperature difference between two surfaces the transmitted radiation from surface i can be defined as:

$$Q_i = \frac{A_i \epsilon_i (\sigma T_i^4 - J_i)}{1 - \epsilon_i} \quad (2)$$

Where:

$Q_i$  = the thermal energy leaving surface i [W]

$\epsilon_i$  = the emissivity of surface i [m<sup>2</sup>]

$\sigma$  =  $5,670373 \cdot 10^{-8}$  [J/s/m<sup>2</sup>/K<sup>4</sup>]

$T_i$  = the temperature of surface i [K]

This means that the radiation flux between two surfaces of different temperature can be expressed by:

$$Q_{ij} = \sigma A_i \epsilon_i (T_i^4 - T_j^4) \quad (3)$$

Where:

$T_i$  = the temperature of surface i [K]

$T_j$  = the temperature of surface j [K]

In COMSOL 5.2 four radiation modules are included. This study focuses on the module: “heat transfer with Surface-to-Surface radiation (ht)” (from: Physics -> Heat Transfer -> Radiation -> heat transfer with Surface-to-Surface radiation(ht)). In the application area of surface to surface radiation in the built environment, the constraints for  $\alpha = \epsilon$ , and constant radiative properties are satisfied. As can be seen from equation (1) the view factor (F) plays an important role within the radiation transfer. The view factor is only dependent of the geometry of the radiating bodies and can be seen as the fraction of radiation from surface i that is intercepted by surface j. It can be expressed as:

$$F_{ij} = \frac{\text{radiation leaving } A_i \text{ and hitting } A_j}{\text{total radiation leaving } A_i} \quad (4)$$

In COMSOL the surface is often divided into a large number of sub-surfaces by meshing the geometry, these sub-surfaces are referred to as facets. The view factor is automatically determined in COMSOL for each of the facets using a “hemicube” or “direct area integration” approach. The view factors can also be extracted from COMSOL (as discussed in the tutorial: view factor computation). However in this version of COMSOL (5.2) there is no functionality to manually input the view factors for the calculation (in case these are known in advance).

## 2. Using Surface-to-Surface radiation

This guide assumes that a geometry (2D / 3D) is present, either with or without physics modules, studies and meshes. Some steps can be skipped, depending on the features already present in the model. To enter radiative heat transfer to the model in COMSOL Multiphysics 5.0, the following steps need to be taken:

1) *Modules*: Add the physics module: Heat Transfer with Surface-to-Surface Radiation (ht) [From: Heat Transfer > Radiation]

2) *Studies*: If no study is present, add the appropriate study (Steady-state / Time Dependent)

3) *Properties*: Add the material properties for conduction in solids (Heat capacity [ $C_p$ ], Density [ $\rho$ ], Conductivity [ $\kappa$ ]) : Either from the materials definition (assigning materials to the geometry) or by selecting “Heat transfer in solids” and choosing user-defined values in case no materials are defined.

4) *Initial values*: Select initial temperature values for the geometries [From: Heat Transfer with Surface-to-Surface Radiation (ht) > Initial values]. This is especially important for time-dependent studies, as this is the starting value for the first time step.

5) *Boundary conditions*: Add thermal boundaries to the geometry (Heat fluxes, Thermal boundaries, Thermal insulation) [Physics > Boundaries > (condition)]

6) *Radiation boundary conditions*: Add surface properties for radiation to the geometry [Physics > Boundaries > Diffuse Surface]. Select a diffuse boundary for each geometry participating in radiation and assign the emissivity value for the surface. This can either be a user-defined value or a value allocated from earlier defined materials. Secondly, make sure that in the Diffuse Surface boundary, the “Include surface-to-surface radiation” option is checked, for each surface you want to include in the radiation study. In addition, define the ambient temperature  $T_{amb}$  for the boundary.

After defining the properties of the diffuse surface, the radiation settings need to be set up. The radiation direction is “opacity controlled” by default. This is applicable for all diffuse surfaces. For non-diffuse or transparent surfaces such as glass, the direction and wavelength-dependence needs to be investigated. The other options include: positive normal direction, negative normal direction and both sides which specifies the direction in which radiation can occur. Important note, each surface which needs to be included within the surface to surface radiation (calculation) needs to have an emissivity value defined! For surfaces which are not defined, COMSOL will use by default an emissivity of 0. This means that this surface will not be affected by radiation. If a certain surface should act as a black body radiator, an emissivity value of 1 needs to be used. A second important note is the definition of  $T_{amb}$  for diffuse surfaces.  $T_{amb}$  can be set to define far-away temperatures in directions where no other boundaries obstruct the view, such as radiation to the sky. Inside a closed volume, the ambient view factor,  $F_{amb}$ , is theoretically zero and the value of  $T_{amb}$  therefore should not matter. However, if walls are defined as a 2D surface, radiation can still take place on the other side of the geometry, influencing temperature results. This can be avoided by adjusting the radiation direction, to only emit inside the volume, or by setting  $T_{amb}$  equal to the indoor (air) temperature.

7) *Meshing*: If no mesh is present, create a mesh for the geometry. COMSOL uses the mesh facets to calculate the local view factors. For large surfaces, a course mesh is often sufficient; however, small discrepancies can be found at sharp corners or at surface edges. This is illustrated in figure 1, where course meshes are compared to fine meshes [1]. If the radiation is calculated from a temperature boundary with a non-uniform distribution, an exact temperature distribution at the radiating boundary is required to calculate the correct radiation fluxes. (This is often the case in models in which convection and radiation modules are combined). “The advancing front” meshing method for the “free triangular” setting can be used to get a dense mesh distribution at the boundary.

8) *Post-processing*: After solving the model [model > compute] the radiation properties can be derived from the model. From the results, line integration profiles for e.g. temperatures, heat fluxes and surface radiosity can be obtained using expressions [Derived values > Line integration > Expression > Expression]. E.g. to define the boundary heat fluxes: [Replace expression > Component > Heat transfer with surface to surface radiation > Boundary fluxes > ht.ndflux - Normal conductive heat flux].

### 3. Benchmarks

This section provides a brief description for different surface-to-surface radiation case studies. Each case focuses on a different configurations related to radiation heat transfer. The modeled results are compared to the analytical solution. Of these analytical solutions, the calculations are added in annex. The modeling files for each of these case are included in the folder. However, meshes and solutions have been cleared from these models to keep the file-size acceptable, and these have to be generated before the (correct) results can be displayed.

#### 3.1 Radiation in a triangular cavity

This case study is used by COMSOL to introduce people to the modeling of surface to surface radiation. Note that the exact guide for this tutorial can be found at their website [4]. Also note that [1] also discusses this case study. This geometry consists of three rectangles (3, 4 and 5 m and a width of 1 m) with the material properties of copper. The rectangles can only exchange heat through surface-to-surface radiation to one another. Input: Heat flux and temperature boundaries, Output: surface-to-surface radiation flux and surface temperatures

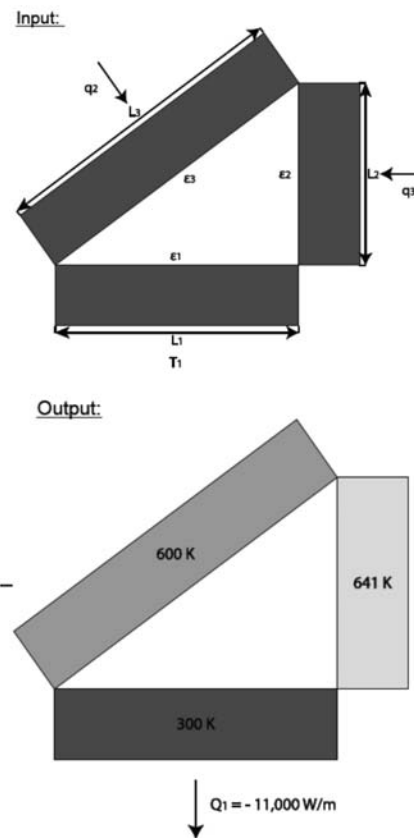


Figure 1. Benchmark 1, Top: Input, Bottom: Output

Table 1. Benchmark 1 numbers

Model outputs	in- & Unit	Object 1	Object 2	Object 3
Temperature T	K	300	641*	600*
Size (L x W)	m	4 x 1	3 x 1	5 x 1
Emissivity $\epsilon$	-	0,4	0,6	0,8
Heat flux Q	W/m <sup>2</sup> W/m	-	2000	1000
		11,000*		

Parameter	Analytical	COMSOL Multiphysics	Difference [%]
Temperature Object 2 [K]	641.00	644.73	0.58
Temperature Object 3 [K]	600.00	600.86	0.14
Heat flux [W/m]	11,000	10,966	0.31

### 3.2 Radiation between two plates

This geometry consists of two rectangles of infinite length and width (modeled in 2D: length 1000 m, thickness 1 m, assumed an infinite width). The rectangles have prescribed temperatures. Input: Heat flux and temperature boundaries, Output: surface-to-surface radiation fluxes and solid temperatures (displayed in the table below).

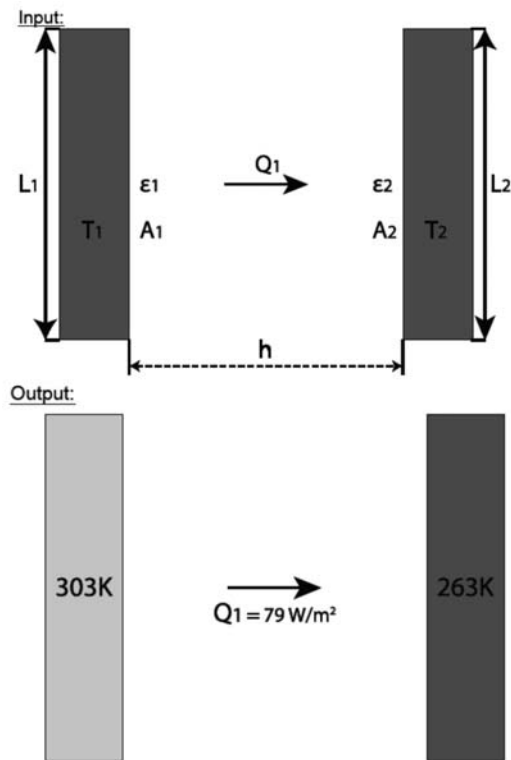


Figure 2. Benchmark 2, Top: Input, Bottom: Output

Table 2. Benchmark 2 numbers

	Unit	Object 1	Object 2
Temperature T	K	303,15	263,15
Size (L x W x H)	m	Infinite*	Infinite*
Distance surface to surface h	m	N.A.	N.A.
Emissivity ε	-	0,9	0,4
Heat flux Q <sub>12</sub>	W/m <sup>2</sup>	<b>79.28*</b>	

Parameter	Analytical	COMSOL Multiphysics	Difference [%]
Heat flux [W/m <sup>2</sup> ]	79.27	79.25	0.02

### 3.3 Radiation in a three dimensional rectangular enclosure

This geometry consists of a 1 x 1 x 1 m rectangular box.

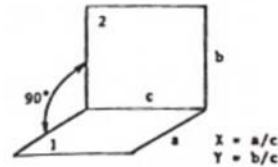


Figure 3. Benchmark 3, Input

Two (non-opposing) walls have a temperature boundary condition applied to them, while the other sides consist of black body radiators ( $\epsilon = 1$ ). Input: temperature boundaries, Output: surface-to-surface radiation fluxes and wall temperatures. The hot- and cold wall input temperatures are 293.15 K and 323.15 K. To model the walls of the box, the COMSOL model consists of a rectangular object for each side, to clearly separate the boundaries

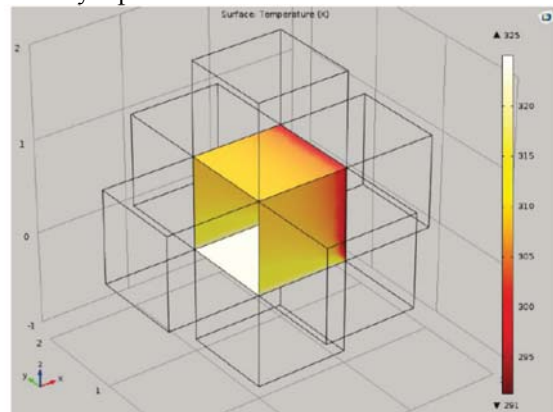


Figure 4. Benchmark 3, Output (Temperatures)

Table 3. Benchmark 3 numbers

	Unit	Hot wall	Cold wall	Black body surfaces
Temperature T	K	322.15	293.35	308.44
Size (L x W)	m	1 x 1	1 x 1	1 x 1
Emissivity ε	-	0.5	0.5	1.0
Heat flux	W	<b>-53.75*</b>	<b>53.75*</b>	N.A

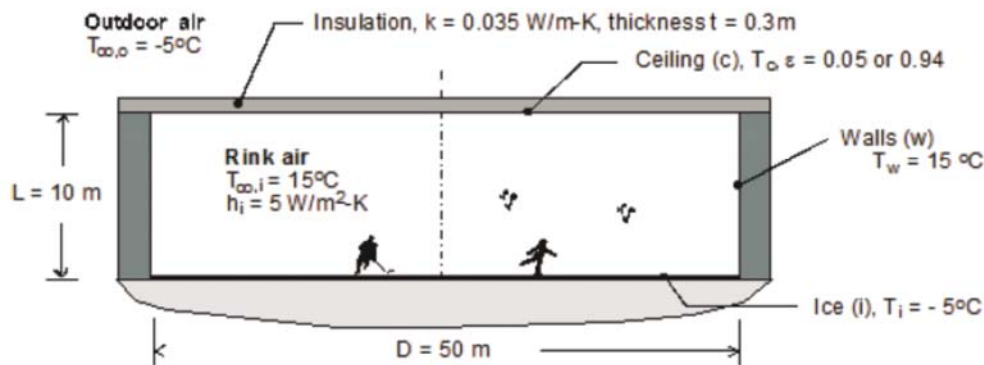
  

Parameter	Analytical	COMSOL Multiphysics	Difference [%]
Black body surface temp [K]:	309.24	308.44	0.27
Heat flux [W]	54	53.75	0.46

### 3.4 Radiation in an ice rink building

This geometry consists of a 50 by 10 m 2D ice-skate hall.

#### SCHEMATIC:



Two side walls have a temperature boundary condition of 15°C. The ice surface has a temperature boundary of -5 °C. The ceiling is a grey body with two possible paint layers ( $\epsilon = 0.05$  or  $\epsilon = 0.94$ ), while the other sides consist of black-body radiators ( $\epsilon = 1$ ). A convective heat flux of 5 W/m<sup>2</sup>K was applied to the lower ceiling boundary to account for convective losses to the air. Input: temperature boundaries/ convective heat-fluxes, surface properties Output: surface temperatures. Two simulations were performed for this case, one for each of the two paint layers: High reflectivity, the ceiling emissivity  $\epsilon = 0.05$ . Low reflectivity, the ceiling emissivity  $\epsilon = 0.94$ .

**Table 4.** Benchmark 4 numbers

Parameter	Analytical	COMSOL Multiphysics	Difference [%]
<b>High reflectivity case:</b>			
Black body surface temp [K]:	287.2	287.0	0.06
<b>Low reflectivity case:</b>			
Black body surface temp [K]:	281.7	280.4	0.46

### 4. Conclusions

The deviation values are calculated based on the absolute values. This allows for a honest comparison between the high and low temperature cases. However, for cases with small temperature differences in [K], the accuracy might be much lower for comparing temperature differences.

Radiative fluxes are very accurately estimated by COMSOL as becomes clear from the validation cases. For the temperatures; however, accuracy is significantly lower. There can be a few reasons why this is the case: the analytical solutions compute an average value for an entire boundary, while COMSOL calculates location specific values. In addition, the effects in sharp corners, as explained in chapter 2, also causes some differences for the area integrated averages. The difference in temperature predictions was most noticeable in the ice rink case (case 3) where estimations for the low reflectivity case, were significantly more accurate than for the high reflectivity case. This would indicate that the more uniform temperature distributions for the low  $\epsilon$  is easier to predict than the curved temperature profile encountered for the high  $\epsilon$  case. This should be taken into account when modeling surface to surface radiation using COMSOL Multiphysics.

Nevertheless, in cases where indoor environments are modeled the radiative heat transfer is often only one aspect of heat transfer, as convective and conductive heat transfer are involved as well. Including these will likely lead to even more precise predictions of building surface temperatures.

## 5. References

- [1] Emery, A. F., Cochran, R. J., Dillon, H., & Mescher, A. (2009). Validation of radiation computations using viewfactors and COMSOL's hemicube approaches. In *Proceedings of the 2009 COMSOL users conference*. Boston
- [2] Bejan, A., & Kraus, A. D. (2003). *Heat transfer handbook* (Vol. 1). John Wiley & Sons.
- [3] Research and Education Association, Fogiel M. (1999), *The Physics Problem Solver Heat transfer, a complete solution guide to any textbook* (Vol. 3). Research & Education Association
- [4] COMSOL Multiphysics, Tutorial- Radiation in a cavity
- [5] COMSOL Multiphysics, Heat transfer module User's Guide
- [6] Radiation exchange between surfaces: Enclosures with non-participating media, Retrieved November 2015
- [7] Eck, R. van & Klep, M. (2016), COMSOL Multiphysics: Surface-to-Surface Radiation Modeling Guide, 7LS4M0 Report, Eindhoven University of Technology.
- [8] HAMLab homepage (visited June 2016). <http://archbpsl.campus.tue.nl/bpswiki/index.php/Hamlab>