Ray Optics Module

User's Guide

5.4

COMSOL

Ray Optics Module User's Guide

Version: COMSOL 5.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM024201

Contents

Chapter 1: Introduction

Chapter 2: Ray Optics Modeling

Chapter 3: Ray Optics Interfaces

Chapter 4: Multiphysics Interfaces and Couplings

Chapter 5: Glossary

1

Introduction

This guide describes the Ray Optics Module, an optional add-on package for COMSOL Multiphysics®.

This chapter introduces you to the capabilities of this module. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

In this chapter:

- About the Ray Optics Module
- Overview of the User's Guide

About the Ray Optics Module

These topics are included in this section:

- The Ray Optics Module Physics Interface Guide
- Common Physics Interface and Feature Settings and Nodes
- Where Do I Access the Documentation and Application Libraries?

The Ray Optics Module Physics Interface Guide

The Ray Optics Module extends the functionality of the physics interfaces of the base package for COMSOL Multiphysics. The details of the physics interfaces and study types for the Ray Optics Module are listed in the table. The functionality of the COMSOL Multiphysics base package is given in the *COMSOL Multiphysics Reference Manual*.

In the *COMSOL Multiphysics Reference Manual*:

- Studies and Solvers
- The Physics Interfaces
- For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Optics				
Ray Optics				
Geometrical Optics		gop	3D, 2D, 2D axisymmetric	ray tracing; bidirectionally coupled ray tracing; time dependent
Ray Heating		—	3D, 2D, 2D axisymmetric	ray tracing; bidirectionally coupled ray tracing; time dependent

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

!	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

!	If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

Win

To open the **Help** window:

- In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1.
- On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1.
- From the **File** menu, click **Help** (?).
- In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac

Linux

To open the **Help** window:

- In the **Model Builder** or **Physics Builder** click a node or window and then press F1.
- On the main toolbar, click the **Help** (?) button.
- From the main menu, select **Help>Help**.

Opening the Documentation Window

Win

To open the **Documentation** window:

- Press Ctrl+F1.
- From the **File** menu select **Help>Documentation** (📘).

Mac

Linux

To open the **Documentation** window:

- Press Ctrl+F1.
- On the main toolbar, click the **Documentation** (📘) button.
- From the main menu, select **Help>Documentation**.

The Application Libraries Window

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or app. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.

The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window

To open the **Application Libraries** window ( ):

Win

- From the **Home** toolbar, **Windows** menu, click ( ) **Applications Libraries**.
- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select ( ) **Update COMSOL Application Library**.

Mac

Linux

Select **Application Libraries** from the main **File>** or **Windows>** menus.

To include the latest versions of model examples, from the **Help** menu select ( ) **Update COMSOL Application Library**.

Contacting COMSOL by Email

For general product information, contact COMSOL at info@comsol.com.

COMSOL Access and Technical Support

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *Ray Optics Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions how to use COMSOL in general are included with the *COMSOL Multiphysics Reference Manual*.

As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS AND INDEX

To help you navigate through this guide, see the Contents and Index.

RAY OPTICS MODELING

The Ray Optics Modeling chapter provides an overview of ray tracing simulation. It begins with an overview of the Essentials of Ray Tracing. It then describes the major functionality groups that are included in The Geometrical Optics Interface, including topics such as Geometry and Meshing, Ray Release Features, Analysis Types, and Results Analysis and Visualization.

RAY OPTICS INTERFACES

The Ray Optics Interfaces chapter describes The Geometrical Optics Interface and includes the Theory for the Geometrical Optics Interface.

MULTIPHYSICS INTERFACES AND COUPLINGS

The Multiphysics Interfaces and Couplings chapter describes The Ray Heating Interface, a dedicated Multiphysics interface for computing heat sources generated by attenuation of rays in absorbing media, and includes the Theory for the Ray Heating Interface.

2

Ray Optics Modeling

This chapter gives an overview of the most important considerations when creating a ray optics model. The most significant modeling decisions include the means of geometry setup, the choice of optical material properties, ray releases, and handling of ray interactions with surfaces in the geometry.

In this chapter:

- Essentials of Ray Tracing
- Geometry and Meshing
- Boundary Conditions
- Ray Release Features
- Additional Variables Solved For
- Analysis Types
- Results Analysis and Visualization
- Variables and Component Couplings

Essentials of Ray Tracing

The Geometrical Optics interface can be used for ray optics simulation of cameras, telescopes, spectrometers, solar collectors, laser systems, room lighting, and much more. However, certain essential elements of the ray optics approach are common to all of these application areas. These universal concepts in ray optics simulation are outlined below.

In this section:

- The Ray Tracing Algorithm
- Basic Requirements of a Ray Optics Model

The Ray Tracing Algorithm

The Geometrical Optics interface solves for the position and wave vector of individual rays. It also allows them to interact with boundaries that intersect their paths.

Ray tracing is usually a reasonable approach to model electromagnetic wave propagation as long as the wavelength of the radiation is small compared to the smallest geometric detail in the surroundings, since diffraction is ignored. As long as this criterion is met, ray tracing can be used for nearly any part of the electromagnetic spectrum, including radio waves, microwaves, visible light, and UV radiation.

Notably, x-rays have such a small wavelength that they can interact with matter on an atomic level. X-ray diffraction, for example, cannot be modeled with a ray tracing approach.

While propagating through a homogeneous medium (one in which the refractive index is spatially uniform), a ray simply goes in a straight line at speed c/n, where $c = 299{,}792{,}458\ \mathrm{m/s}$ is the speed of light in a vacuum and n (dimensionless) is the absolute refractive index of the medium. In a graded-index medium, the ray can follow a curved path, which is determined by integrating coupled first-order ordinary differential equations over time.

To learn more about the equations of ray propagation and their derivation, see Introduction to Geometrical Optics in the Theory for the Geometrical Optics Interface chapter.

The Geometrical Optics interface is compatible with the **Ray Tracing** and **Time Dependent** study steps. These study types are very similar, except that the **Ray Tracing** study step allows you to specify a number of optical path length intervals instead of time steps. (Internally, the optical path lengths are converted to the corresponding times, so this is just a matter of convenience.)

For simple ray tracing models, only the first and last time steps might be needed. Then the behavior of rays at any intermediate time can be accurately interpolated. If rays interact with boundaries in-between the stored time steps, then the exact time and position of each ray-boundary interaction is also stored and readily available.

Finer time stepping may be needed when modeling ray propagation in graded-index media, when rays pass through attenuating media and generate heat in their surroundings, or when using certain specialized postprocessing features.

Every time the intersection of a ray with a surface is detected, any one of a wide variety of ray-boundary interactions may apply, including specular reflection, diffuse reflection, refraction, and several different types of absorption.

Along each ray, it is possible to evaluate expressions that involve variables defined on the ray itself (such as optical path length, intensity, and wavelength) and variables defined at the ray's position in the modeling domain (such as temperature and refractive index). For example, to determine the refractive index in an optically dispersive medium, on each ray an expression is evaluated that combines the ray's wavelength or frequency with a function queried from the domain the ray occupies. Similarly, when a ray hits a boundary, the new direction of the ray can depend on a combination of ray variables (like wavelength) and surface variables (like surface normal direction and Gaussian curvature).

Basic Requirements of a Ray Optics Model

Although geometrical optics models vary tremendously in application area and scope, every geometrical optics model requires at least the following basic features:

- Some rays must be released into the model. This requires at least one r*ay release feature,* such as the Release, Inlet, or Release from Grid node. See Ray Release Features for more details.
- Some geometry must be present, with at least one surface that can interact with the rays. Thus, the model must have at least one *Boundary condition*, such as the Material Discontinuity or Wall node. By default, every boundary adjacent to at least

one domain in the model is treated as a **Material Discontinuity** that can reflect and refract rays. See Boundary Conditions for more information.

- Every model includes the Ray Properties node, a default node that cannot be removed. This node defines the equations of ray propagation. If the radiation is monochromatic, this is also where the frequency or wavelength is specified.
- The refractive index must be specified. If any domains are present, use the Medium Properties node to either specify the refractive index directly or specify which material controls it.
- The ray tracing algorithm detects ray-boundary interactions using the underlying finite element mesh. At the very least, a surface mesh is needed. For multiphysics modeling in which field variables like temperature are solved for, a domain mesh is also necessary. Some related topics are discussed in the Geometry and Meshing. section.
- A study is necessary to compute the ray paths. The Geometrical Optics interface is compatible with the **Time Dependent**, **Ray Tracing**, and **Bidirectionally Coupled Ray Tracing** study steps. **Ray Tracing** is recommended for most models. See Analysis Types for more information.

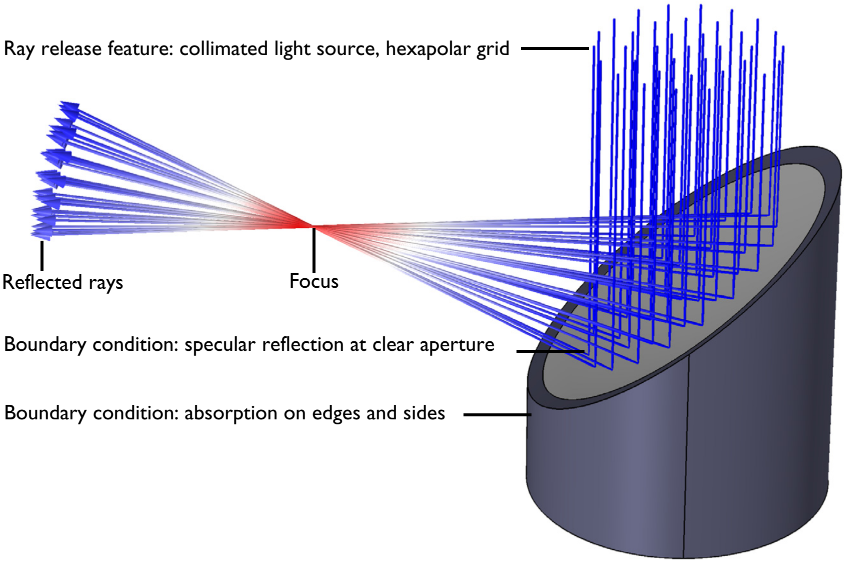

Ray release feature: collimated light source, hexapolar grid
Reflected rays
Focus
Boundary condition: specular reflection at clear aperture
Boundary condition: absorption on edges and sides

Figure 2-1: A simple ray tracing model, featuring specular reflection of a collimated beam by an off-axis conic mirror. The color expression indicates the ray intensity.

Geometry and Meshing

Electromagnetic radiation can interact with a wide variety of geometric entities. In ray optics simulation, the most common elements include lenses, mirrors, prisms, beam splitters, light pipes, and various obstructions. Rays may also interact with buildings, vehicles, people, planets, and more.

The Geometrical Optics interface supports interaction with any type of geometric entity. The geometry can be constructed from primitives like circles and line segments, loaded from various CAD formats, or constructed with the aid of parts from the Ray Optics Part Library. Rays can also interact with both deformed and undeformed geometries, for example, when an object undergoes thermal expansion.

In this section, various considerations for setting up the model geometry and the associated finite element mesh are considered in more detail.

In this section:

Domain Selection

It is possible for rays to pass through domains in the geometry and to propagate in the void region outside these domains. Boundary conditions can be specified at any boundary, even at boundaries that are not adjacent to any domain in the geometry. This means that a ray can be reflected or absorbed by a surface in 3D or a line segment in 2D even if it isn't attached to any other object. The ray tracing algorithm can also detect boundary interactions in any order, without this order being specified.

In the physics interface **Ray Release and Propagation** section, the **Refractive index of exterior domains** is specified. This refractive index is used in any domains outside of the selection for the Geometrical Optics interface, as well as the void domain outside the geometry. It is a constant, scalar-valued quantity; thus, the refractive index outside the domain selection cannot depend on field variables such as temperature and cannot be a graded-index medium. The default refractive index of 1 represents a perfect vacuum.

Usually, the domain selection for the Geometrical Optics interface should include all objects that the rays might pass through. In a lens system, this would mean all lenses are included, but not necessarily the mount for these lenses. However, rays can also pass through the void region that surrounds the geometry.

A major advantage to excluding some domains from the selection for the physics interface is that these domains do not need to be meshed. However, these domains would still require a mesh if some other variables, like displacement and temperature, are solved for there. See Meshing Guidelines for Geometrical Optics Simulation for more details.

Note that some physics features require a domain mesh and will not function on domains outside the physics interface selection. This includes all types of Accumulator (Domain) feature, including the dedicated Ray Heat Source multiphysics feature.

Ray Tracing in an Imported Mesh

It is possible to compute ray trajectories in an imported mesh. The mesh can be imported from a COMSOL Multiphysics file (`.mphbin` for a binary file format or `.mphtxt` for a text file format) or from a NASTRAN file (`.nas`, `.bdf`, `.nastran`, or `.dat`).

If the mesh is imported from a COMSOL Multiphysics file, the imported mesh always uses linear geometry shape order for the purpose of modeling ray-boundary interactions, even if the model used to generate the mesh had a higher geometry shape order.

If the mesh is imported from a NASTRAN file, the ray-boundary interactions may be modeled using either linear or higher geometry shape order. If **Export as linear elements** is selected when generating the NASTRAN file, or if **Import as linear elements** is selected when importing the file, then linear geometry shape order will be used.

Part Libraries

In geometrical optics simulation, it is often necessary to set up geometry sequences consisting of entities that are more complex than simple geometry primitives such as spheres, cones, blocks, etc. Instead it might be necessary to insert lenses with different thicknesses and radii of curvature, off-axis conic mirrors, and parabolic concentrators. This can be conveniently accomplished using the Part Library for the Ray Optics Module.

The Part Library contains fully parameterized sequences of geometry instructions that produce more complex shapes frequently required for geometrical optics simulation, including the following:

- Spherical lenses
- Cylindrical lenses
- Beam splitters
- Parabolic mirrors
- Prisms
- Retroreflectors

For example, you can load the *Spherical Lens 3D* part into a model and then specify the radii of curvature of each lens surface, along with the lens thickness and diameter. Ray Optics parts can include cumulative selections that make it easy to apply a boundary condition to a group of lens surfaces at the same time.

In addition, many parts automatically define work planes so that the parts and other features can more easily be positioned and oriented with respect to each other.

Many Ray Optics tutorials use the Part Library to create their geometry sequences. To learn more, see the following models:

- *Newtonian Telescope*: Application Library path **Ray_Optics_Module/ Lenses_Cameras_and_Telescopes/newtonian_telescope**
- *Petzval Lens*: Application Library path **Ray_Optics_Module/ Lenses_Cameras_and_Telescopes/petzval_lens**

Part Libraries in the *COMSOL Multiphysics Reference Manual*.

Meshing and Discretization Error

In the Geometrical Optics interface, rays don’t interact with an exact analytical representation of the geometry, but rather they interact with the underlying finite element mesh. For example, when detecting ray-boundary interactions including reflection, refraction, and absorption, the intersection point of each ray with a surface is actually the intersection point with a boundary element on that surface.

The advantage of using a mesh representation of the geometry to detect and apply ray-boundary interactions is that the algorithm is readily extended to high-fidelity multiphysics simulation including translational motion, rotation, and structural deformation including thermal stress. In addition, this implementation allows rays to be traced through geometric entities of arbitrary shape, not just simple shapes for which a parametric representation is readily available.

To ensure that ray reflections and refractions are both detected and applied accurately, the mesh must be of sufficiently high quality. This is trivial when the surfaces are planar because even a small number of linear boundary elements can represent a planar surface to machine precision. Accurately discretizing the geometry becomes more important when the surfaces are curved, as in spherical lenses and conic mirrors, or when the surfaces may be deformed.

In the following image, the radial position over one boundary element on a coarsely meshed sphere (using quadratic elements) is compared to the exact value for an ideal sphere. The error is on the order of 10^{-5}.

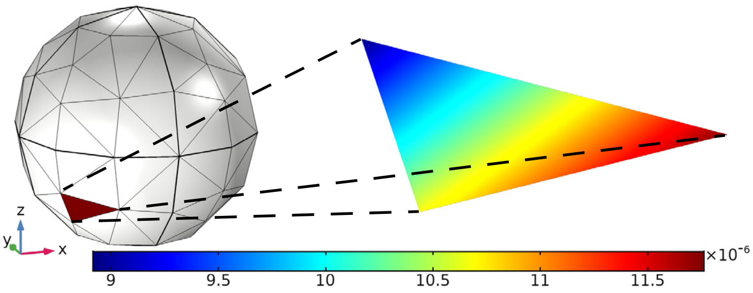


Figure 2-2: Error in the radial displacement over one boundary element of a unit sphere.

The discretization error not only applies to the position on the sphere, but also to the normal direction and Gaussian curvatures of the surface, all of which can be used in geometrical optics simulation. A relative error like the one shown above (10^{-5}) might be sufficiently small for some simulation results, but in geometrical optics such an error might translate to tens of additional wavelengths in spot size—large enough to invalidate the results of the simulation entirely, unless adequate precautions are taken.

Similarly, in models with mesh deformation, the degrees of freedom for the displacement field must be solved for extremely accurately for the results of a coupled multiphysics model to be trusted. A good practice, as with other types of simulation, is to perform a mesh refinement study, ensuring that the results don't change appreciably when the mesh element size is reduced further.

Meshing Guidelines for Geometrical Optics Simulation

- Flat edges (2D) and planar surfaces (3D) can be coarsely meshed, unless they meet one of the criteria described below.
- Curved surfaces that can interact with rays should always be finely meshed. The tighter the curvature of such surfaces, the finer the mesh should be.
- Domains usually don't need to be as finely meshed as boundaries. A convenient way to refine the mesh in the vicinity of the boundaries is to reduce the **Curvature factor** in the **Size** settings window in the mesh sequence. This results in a finer mesh only where the radius of curvature of the surface is small. You might also have to reduce the **Minimum element size** to avoid warnings in the mesh sequence.
- Features that compute the density of some quantity on a domain or boundary usually require a finer mesh, because the density term is piecewise discontinuous across elements. This includes the Accumulator (Boundary), Accumulator (Domain), and Deposited Ray Power (Boundary) features. If the mesh is too fine, rays might entirely miss some elements, and then it is necessary to increase the number of rays to avoid "holes" in the deposited power or other density field.
- Avoid unnecessary edges when possible. Discretization error tends to be larger in mesh elements touching geometry edges.
- Whenever possible, use the geometry Parts from the Part Library for the Ray Optics Module, rather than creating lenses from geometry primitives like the sphere and cylinder. The built-in lens parts have been constructed in a specific way so as to reduce the discretization error on the reflecting and refracting surfaces.
- Consider changing the **Geometry shape order** in the model component settings. Using **Cubic** or **Quartic** causes the boundaries to be discretized using higher-order polynomials, which can reduce error by several orders of magnitude.

  Similarly, with physics interfaces that solve for a displacement field, such as Solid Mechanics, locate the physics interface **Discretization** section. A higher shape order such as **Cubic Lagrange** should be selected if rays are traced in the deformed geometry
- If the geometry uses parts or primitives, you can reduce discretization error by selecting **Use geometry normals for ray-boundary interactions** in the physics interface **Ray Release and Propagation** section. However, this only improves the accuracy if the geometry is undeformed; it has no effect if the geometry is subjected to thermal stress or some other type of deformation.

Boundary Conditions

This section describes the boundary conditions of the Geometrical Optics interface in greater detail.

In this section:

- Reflection and Refraction
- Primary and Secondary Ray Releases
- Diffuse and Specular Reflection
- Absorption
- Special Boundary Conditions

Reflection and Refraction

The default boundary condition is the Material Discontinuity condition on all interior and exterior boundaries. The **Material Discontinuity** causes rays to be reflected and refracted if the two adjacent domains have different refractive indices.

The direction of the refracted ray is based on Snell's law. If the ray intensity or power is solved for, then they are reinitialized according to the Fresnel equations. You can modify the application of Fresnel equations by adding one or more thin dielectric layers to the surface.

Material Discontinuity Theory and Intensity, Wavefront Curvature, and Polarization in Theory for the Geometrical Optics Interface.

SUPPRESSING THE RELEASE OF REFLECTED RAYS

The total number of released secondary rays in a model can sometimes grow rapidly and exhaust all of the preallocated secondary degrees of freedom. For example, a single ray reflecting back and forth between two **Material Discontinuity** boundaries can create an inordinately large number of rays, each with extremely low intensity.

It can be useful to constrain the release of secondary rays at boundaries so that only the most important rays are produced. If reflected rays are not of any interest at all, then in the **Rays to Release** section, select **Never** from the **Release reflected rays** list. If reflected rays are only relevant to the model under a certain condition, such as hitting a specific part of the surface or having a certain direction, instead select **Based on logical expression** and then enter a user-defined **Evaluation expression** that must be satisfied.

If ray intensity is solved for, you can also specify a **Threshold intensity**. If a reflected ray would have intensity below the threshold, it isn't released at all. Similarly, if ray power is solved for, you can specify a **Threshold power**.

Primary and Secondary Ray Releases

During the process of reflection and refraction described in the previous section, interaction with a boundary caused one ray to split into two. Hence, this boundary condition also functions as a ray release feature.

In the Geometrical Optics interface, rays are designated as either *primary* or *secondary*. Similarly, the physics features that produce them are called *primary release features* or *secondary release features*. The above is an example of a secondary release.

PRIMARY RAYS

A primary release feature allows the initial position and direction of rays to be specified directly. For the release positions, either specify the grid points directly (as in the Release from Grid feature) or choose the geometric entities that produce the rays (as in the Release, Inlet, Release from Edge, and Release from Point features). The initial direction can be specified directly or sampled from a distribution.

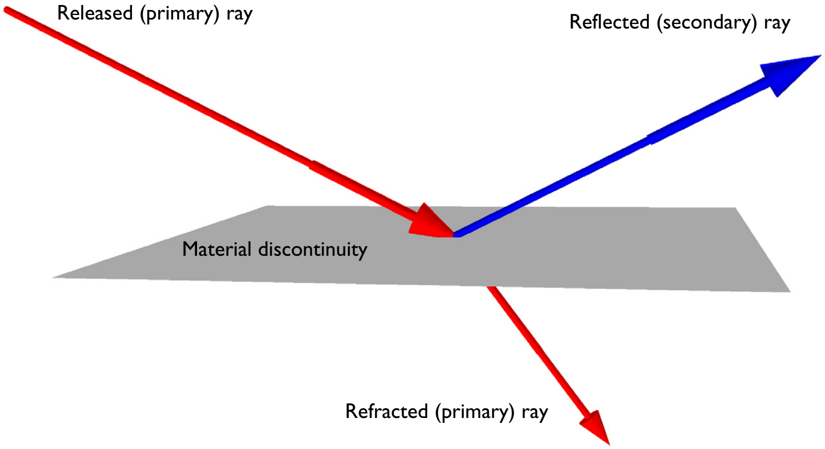
Primary rays are released directly by a release feature. They are called primary rays because their release is not contingent on the prior existence of any other ray.

SECONDARY RAYS

Secondary rays are only released when an existing ray is subjected to certain boundary conditions. This existing ray might be a primary ray, or it could be a different secondary ray that was released earlier in the simulation.

For example, the following diagram shows an incident ray being split into reflected and refracted rays at a Material Discontinuity where the refractive indices on either side differ. The Geometrical Optics interface always applies deterministic ray splitting at such boundaries, so when one ray reaches the surface, two rays emerge from it. The refracted ray is a continuation of the incident ray because it has the same index and uses the same degrees of freedom. The reflected ray is a secondary ray.

Note that total internal reflection is automatically detected, and in this case no secondary emission takes place.



ALLOCATION OF SECONDARY DOFS

The number of degrees of freedom allocated to secondary rays is determined when the study begins, and if this maximum number is ever reached, then no additional secondary rays will be released. In other words, if a large number of reflected rays have been produced at material discontinuities in a model, at some point no more reflected rays will be produced. In this case, a **Warning** will appear in the solver sequence. The maximum number of secondary rays that can be produced in a model is controlled by the **Maximum number of secondary rays** field in the physics interface **Ray Release and Propagation** section.

Similarly, the **Grating** boundary condition can release both reflected and transmitted rays of many diffraction orders. One of these reflected or transmitted rays uses the degrees of freedom of the incident ray, and the rest must be taken from the preallocated secondary degrees of freedom.

Diffuse and Specular Reflection

For most boundary conditions other than refraction, the Wall feature can be used. It includes a wide variety of boundary conditions including the following:

- Specular reflection,
- Diffuse reflection,
- Combination of diffuse and specular reflection,
- User-defined reflection,

- Pass through, and
- Several varieties of absorption condition.

In this context “Diffuse reflection” means Lambertian scattering, following the cosine law.

If you choose **Mixed diffuse and specular reflection** you can assign some probability that the ray is reflected specularly; otherwise it is reflected diffusely.

The **Diffuse scattering** and **Mixed diffuse and specular reflection** rely on pseudorandom number generation, so they are not guaranteed to give exactly the same numeric results in different software versions or on different architectures.

Absorption

There are several different types of absorption condition, classified by the type of information that they retain about the ray.

- The **Disappear** condition annihilates the ray completely. After the ray disappears, its position and other degrees of freedom evaluate to not-a-number (NaN) at any later solution time.
- The **Freeze** condition retains the ray position and direction after the ray hits the boundary, although these quantities no longer change over time.
- The **Stick** condition retains the ray position, but all other degrees of freedom are set to zero.

Special Boundary Conditions

More specialized boundary conditions are available for some application areas.

The Grating boundary condition, along with its Diffraction Order subnodes, can be used to reflect rays and emit secondary rays to model interaction with gratings. You can specify the direction of periodicity and the size of the unit cell.

Czerny-Turner Monochromator: Application Library path **Ray_Optics_Module/Spectrometers_and_Monochromators/czerny_turner_monochromator**

The Mirror boundary condition is a simplified Wall that only causes specular reflection.

The Axial Symmetry boundary condition is only available in 2D axisymmetric models. It is automatically applied at the axis of symmetry and can't be applied anywhere else.

Optical devices like the Linear Polarizer and Linear Wave Retarder are available when ray intensity or power is computed. They don't have any effect on the ray direction but they can affect the ray polarization.

The Wall condition also support condition ray-wall interactions. For example, you could cause rays to **Freeze** if a logical expression is satisfied—for example, having optical path length greater than a specified threshold—and subject them to **Specular reflection** otherwise. Together with the **Mixed diffuse and specular reflection** condition, it is possible to prescribe up to 3 different types of wall interaction at a single boundary.

Ray Release Features

To trace rays it is first necessary to prescribe their initial position and direction. This process is called *releasing rays*, and the physics features used to enter this information are called *release features* or *ray release features*. If other quantities are being solved for along the rays, such as the intensity and polarization, then these quantities are also initialized by the ray release features.

In this section:

- Grid-Based Release
- Release from Domains, Boundaries, Edges, or Points
- Specialized Release Features

Grid-Based Release

Use the Release from Grid feature to specify the initial positions of rays using a grid of points. It is useful to release rays from a grid when the initial ray position is known exactly. A grid-based release may be used, for example, when rays are released from the focus of a lens or when a system is excited by a laser. This is the easiest way to release rays from known locations in the void region outside the geometry.

As an alternative to typing points into a text field, you can load them from a file. The Release from Data File node can be used to import the initial ray positions and directions from a text file.

Release from Domains, Boundaries, Edges, or Points

There are ray release features for every geometric entity level:

- Use the Release feature to release rays from domains in 2D or 3D.
- Use the Inlet feature to release rays from edges in 2D or surfaces in 3D.
- Use the Release from Edge feature to release rays from edges in 3D.
- Use the Release from Point feature to release rays from points in 2D or 3D.

Releasing rays from a domain, surface, or edge initializes the ray position based on the underlying finite element mesh, so the ray positions are affected by the mesh. A side effect is that the ray positions can change slightly when switching between geometry kernels.

Ray release can be uniform or proportional to a user-defined expression, which is specified in the **Density proportional to** text field.

Specialized Release Features

The Illuminated Surface is a specialized ray release feature that produces reflected or refracted light directly at a surface. This can be used, for example, when the direction of incident radiation is known, but its propagation is not interesting until it has already been reflected once by an object.

The Solar Radiation computes the initial direction of rays as if they were solar rays being released at a specified latitude, longitude, date, and time. Instead of the latitude an longitude, you can also select from a list of built-in cities.

The following examples involve buildings or other objects being illuminated by solar radiation in a specific direction:

- *Solar Dish Receiver*: Application Library path **Ray_Optics_Module/Solar_Radiation/solar_dish_receiver**
- *Vdara® Caustic Surface*: Application Library path **Ray_Optics_Module/Solar_Radiation/vdara_caustic_surface**

Additional Variables Solved For

The Geometrical Optics interface always solves for the ray position (denoted **q**) and wave vector (denoted **k**). This section lists the optional quantities that can be solved for in addition to these required variables.

In this section:

- Modeling Polychromatic Radiation
- Intensity, Polarization, and Power
- Other Dependent Variables
- Order of Initialization of Auxiliary Dependent Variables

Modeling Polychromatic Radiation

By default, the rays released in a model are monochromatic with a ray frequency or free-space wavelength defined in the Ray Properties settings window. However, you can trace polychromatic rays by specifying different frequencies or vacuum wavelengths in each ray release feature.

To allow the rays to be polychromatic, in the settings window for the Geometrical Optics interface locate the **Ray Release and Propagation** section. By default, **Monochromatic** is selected from the **Wavelength distribution of released rays** list. Select **Polychromatic, specify vacuum wavelength** to release polychromatic rays by entering an expression for the vacuum wavelength or sampling it from a distribution. Alternatively, select **Polychromatic, specify frequency** to define an expression or distribution for the ray frequency. These expressions are defined in the sections **Vacuum Wavelength** and **Initial Ray Frequency**, respectively, in each ray release feature.

When modeling polychromatic light, the number of degrees of freedom in the model increases by one per ray because the wavelength or frequency is stored as an auxiliary dependent variable on each ray.

Czerny-Turner Monochromator: Application Library path **Ray_Optics_Module/Spectrometers_and_Monochromators/czerny_turner_monochromator**

Intensity, Polarization, and Power

Ray intensity is computed using a variant of the Stokes-Mueller calculus in which both the amplitude and polarization are tracked along individual rays.

LIST OF AVAILABLE SETTINGS

To decide whether intensity is computed, select an option from the **Intensity Computation** list. The following options are available.

- **None**: Does not compute any intensity information.
- **Compute intensity**: Solves for intensity, which typically increases as rays are focused and decreases as they diverge. Also affected by reflection, refraction, and attenuating media. Only valid when the media are homogeneous.
- **Compute power**: Solves for power, which is unaffected by the convergence or divergence of rays but is still affected by reflection, refraction, and attenuating media. Can be used to compute heat source terms in attenuating domains, or heat flux terms on absorbing boundaries the rays hit.
- **Compute intensity and power**: Combines the capabilities of **Compute intensity** and **Compute power**, at the cost of a few extra degrees of freedom per ray.
- **Compute intensity in graded media**: Similar to **Compute intensity**, but is also applicable to graded-index media. The trade-off is that this method is slower and less accurate for homogeneous media.
- **Compute intensity and power in graded media**: Similar to **Compute intensity in graded media**, but can also be used to generate heat sources in attenuating domains and heat flux terms at boundaries.

HANDLING POLARIZATION

Whenever intensity or power is solved for, the polarization of every ray is known. Rays can have any degree of polarization, ranging from 0 (unpolarized) to 1 (fully polarized) and anything in-between. When rays have some degree of polarization, they can be linearly, elliptically, or circularly polarized.

When rays are reflected and refracted at boundaries, the intensity, polarization, and power are updated based on the Fresnel equations, which automatically take the polarization direction into account.

The polarization is determined based on the Stokes parameters, which are allocated as extra degrees of freedom along each ray. For more information, see The Stokes Parameters in the Theory for the Geometrical Optics Interface chapter.

In the following examples, ray polarization is manipulated in an instructive way:

- *Total Internal Reflection Thin-Film Achromatic Phase Shifter (TIRTF APS)*: Application Library path **Ray_Optics_Module/Prisms_and_Coatings/achromatic_phase_shifter**
- *Linear Wave Retarder*: Application Library path **Ray_Optics_Module/Tutorials/linear_wave_retarder**

Wavefront Curvature

When the ray intensity is solved for, it increases where rays are focused together and decreases where rays diverge. This is accomplished by treating each ray as a wavefront and storing its principal radii of curvature as extra degrees of freedom. In this way, all released rays are treated as points on planar, spherical, or ellipsoid-shaped wavefronts.

For more information on wavefront radii of curvature and their effect on intensity, see Principal Radii of Curvature in the Theory for the Geometrical Optics Interface chapter.

Computing Deposited Ray Power

The options **Compute power**, **Compute intensity and power**, and **Compute intensity and power in graded media** all allow heat sources to be defined on domains or boundaries. As rays propagate through an attenuating medium—that is, a medium where the refractive index is complex-valued—some energy is lost from the ray. The corresponding heat source on the surrounding domain can be computed using either the Deposited Ray Power (Boundary) subnode or the Ray Heat Source multiphysics node. A **Ray Heat Source** node is automatically created when using selecting The Ray Heating Interface in the Model Wizard. The heat generated as rays propagate in an attenuating medium can be used to define a heat source in the Heat Transfer in Solids interface or another interface that computes a temperature field.

- *Thermally Induced Focal Shift in High-Power Laser Focusing Systems*: Application Library path **Ray_Optics_Module/Structural_Thermal_Optical_Performance_Analysis/thermally_induced_focal_shift**

TOTAL POWER TRANSMITTED AND REFLECTED AT GRATINGS

The Grating feature is used to model the transmission and reflection of rays at diffraction gratings. It includes a Diffraction Order subnode to specify which diffraction orders to release. When the ray power is solved for, the **Store total transmitted power** and **Store total reflected power** check boxes are shown in the **Grating** settings window. Selecting either of these check boxes causes an auxiliary dependent variable to be declared, storing the total power of the transmitted and reflected rays of all diffraction orders.

Diffraction Grating: Application Library path **Ray_Optics_Module/ Tutorials/diffraction_grating**

Other Dependent Variables

It is possible to define an auxiliary dependent variable for the optical path length by selecting the **Compute optical path length** check box in the **Additional Variables** section of the physics interface node's Settings window. Initially the optical path length is set to 0 for all released rays. It is possible to reset the optical path length to 0 when the rays interact with boundaries.

The phase of a ray is necessary for some applications that require information about the instantaneous electric fields of multiple rays, such as interferometers. To define an auxiliary dependent variable for phase, select the **Compute phase** check box in the **Intensity Computation** section of the physics interface node's Settings window. This check box is only available if the ray intensity is computed.

The instantaneous phase can be used to visualize interference patterns where the rays intersect a surface. See the Results Analysis and Visualization section.

Other dependent variables can also be assigned for more esoteric purposes, including:

- Number times each ray has been reflected
- Help variables for more accurately tracing rays in strongly absorbing media, where the real and imaginary parts of the refractive index are comparable in magnitude
- Help variables used to apply perturbations in ray direction due to rough surfaces
- User-defined Auxiliary Dependent Variable nodes

Order of Initialization of Auxiliary Dependent Variables

When rays are released, the variables defined for each ray are initialized in a specific order. The initial values of ray variables can only depend on the values of variables that have already been defined. The order of dependent variable initialization is governed by the following rules:

- The initial ray position is always determined first.
- By default, user-defined auxiliary dependent variables (that is, those that are defined by adding an Auxiliary Dependent Variable node to the physics interface) are initialized after all other variables. They can instead be initialized immediately after the ray position vector components by selecting the **Initialize before wave vector** check box shown in the release feature **Initial Value of Auxiliary Dependent Variables** section.
- If more than one user-defined auxiliary dependent variable is present, these variables are initialized in the order in which the corresponding **Auxiliary Dependent Variable** nodes appear in the Model Builder.
- The remaining degrees of freedom are defined in the following order (each listed group is initialized simultaneously, and the variables within a group cannot reliably be initialized in terms of each other):
  - Help variable for the perturbation of initial ray direction at illuminated surfaces,
  - Ray frequency or vacuum wavelength,
  - The wave vector components, optical path length, and total power transmitted and reflected by gratings,
  - The integral of the attenuation coefficient along ray paths and the components of the unit vector that indicates the direction corresponding to one of the principal radii of curvature,
  - The principal radii of curvature, initial principal radii of curvature, intensity, Stokes parameters, and help variables for computing the curvature tensor and intensity, and
  - The total power transmitted by the ray.

Items in each bullet point may not be initialized as functions of items in a later bullet point. For example, the initial ray direction vector may depend on the ray frequency, but the initial principal radii of curvature may not depend on the total power transmitted by the ray.

Analysis Types

The Geometrical Optics interface is compatible with three study types: **Ray Tracing**, **Time Dependent**, and **Bidirectionally Coupled Ray Tracing**. In this section some of the more relevant study settings are explored in greater detail.

In this section:

- Using the Ray Tracing Study Step
- Using the Bidirectionally Coupled Ray Tracing Study Step

Using the Ray Tracing Study Step

The **Ray Tracing** and **Time Dependent** study steps are very similar, and either one could be used for the vast majority of geometrical optics models. The **Ray Tracing** study step has some additional features that make it more convenient to use, such as more reasonable default values and built-in stop conditions.

A NOTE ON GEOMETRIC NONLINEARITY

If a physics interface solves for the displacement field, such as the Solid Mechanics interface, then the check box **Include geometric nonlinearity** appears in the **Study Settings** section. It is very important to select this check box when tracing rays in a system that is deformed due to external forces or thermal stress. If the check box is cleared, then rays are instead traced in the undeformed geometry.

TIME STEPS AND OPTICAL PATH LENGTH INTERVALS

By default, the **Ray Tracing** study step computes ray trajectories from $t = 0$ to $t = 1$ ns with a time step size of 0.01 ns. However, it is often useful to think of ray tracing in terms of the maximum distance of ray propagation instead of the maximum time. To express the duration of the study in terms of a maximum optical path length, change the **Time step specification** setting from the default, **Specify time steps**, to **Specify maximum path length**. Then select a **Length unit** (default m), enter a set of **Lengths** (default `range(0,0.01,1)`), and enter a **Characteristic group velocity** (default `c_const`, a built-in constant for the speed of light in a vacuum). With the default solver settings, the time-dependent solver must take at least one time step whenever the optical path length of a ray moving at the **Characteristic group velocity** would have reached one of the values in the list of **Lengths**.

BUILT-IN STOP CONDITIONS

The **Ray Tracing** study step includes options to create a **Stop condition** node in the default solver sequence. The **Stop condition** node can terminate the study before the full range of specified times or optical path lengths has been simulated, if a condition is met before then.

To use one of the built-in stop conditions, select one of the following options from the **Stop condition** list in the **Study Settings**:

- **None**: The study ends at the specified maximum time or maximum optical path length.
- **No active rays remaining**: The study terminates if all rays have been stopped or absorbed. A ray can be absorbed by a boundary, or it can be removed by the Ray Termination domain feature.
- **Active rays have intensity below threshold**: This option is only available when ray intensity is computed. The study terminates if every ray in the model is either stopped, absorbed, or has sufficiently low intensity.
- **Active rays exceed maximum number of reflections**: This option is only usable when the **Count reflections** check box is selected in the physics interface **Additional Variables** section. The study terminates if every ray in the model is either stopped, absorbed, or has been reflected the specified number of times.

Stop Condition in the *COMSOL Multiphysics Reference Manual*.

COUPLED PHYSICS INTERFACES

If other physics interfaces are also solved for in the **Ray Tracing** study step, it is assumed that these other fields vary over the same time scale as the ray propagation. This is seldom true. If instead the coupled physics do not change over the time scale for ray propagation, consider a study with two steps: **Stationary** for all other fields and **Ray Tracing** just for the rays.

In the *COMSOL Multiphysics Reference Manual*:

- Ray Tracing
- Studies and Solvers

Using the Bidirectionally Coupled Ray Tracing Study Step

The **Bidirectionally coupled ray tracing** study step is a dedicated study step for ray heating and similar applications.

It should only be used if all of the following criteria are met:

1. Rays are being traced.
2. Some other field, such as temperature or structural displacement, is solved for in a domain where the rays are being traced.
3. All fields, apart from the ray paths themselves, are stationary.
4. The ray paths are affected by the field being solved for. This could include rays interacting with a deformed geometry, or a refractive index that depends on the values of field variables like strain or temperature.
5. The rays generate enough heat to significantly affect one of the fields being solved for in the domain, usually temperature.

If condition 4 isn't satisfied, instead use a **Stationary** study step for the fields, followed by a **Ray Tracing** study step for the rays.

In addition to the settings that are available for the **Ray tracing** study step, it is possible to specify a **Number of iterations**. The default value is 5. If the **Bidirectionally coupled ray tracing** study step is used with The Ray Heating Interface, the following iterative solver loop is automatically set up to compute the ray trajectories and temperature:

1. Solve for the temperature field, assuming that the rays do not generate any heat source.
2. Using the temperature computed during the previous step, compute the ray paths and any heat sources that occur due to ray attenuation in an absorbing medium.
3. Using the heat source computed in the previous step, compute the temperature field.
4. Alternate between steps 2 and 3 for the specified **Number of iterations**, or specify a **Global variable** whose convergence will be used to terminate the loop.

The result of the iterative solver loop is that the heat source generated by the attenuation of rays is taken into account when computing the temperature. Through the thermo-optic dispersion model, the temperature in turn affects the ray paths. Thus, a bidirectional coupling is established between the two physics interfaces.

It is possible to extend this bidirectional coupling to include other physical effects. For example, to include structural deformation due to thermal stress, add the Solid

Mechanics physics interface and the **Thermal Expansion** and **Temperature Coupling** Multiphysics couplings.

In the *COMSOL Multiphysics Reference Manual*:

- Bidirectionally Coupled Ray Tracing
- Studies and Solvers

Results Analysis and Visualization

The results of a ray tracing simulation can be interpreted and visualized using a variety of built-in features. In the following, some dedicated features are discussed in detail.

In this section:

- Ray Trajectories Plot
- Ray Plot
- Interference Pattern Plot
- Poincaré Maps and Phase Portraits
- Computing Monochromatic Aberrations

Ray Trajectories Plot

The **Ray Trajectories** plot is a default plot that shows each ray as a line in 2D or 3D. The final ray position can be rendered as a point or vector. It is also possible to modify the **Ray Trajectories** plot by adding the **Color Expression**, **Deformation**, or **Filter** attributes.

When the ray intensity is computed, you can also plot polarization ellipses along the ray paths, using the built-in variables (`gop.pax`, `gop.pay`,`gop.paz`) for the semi-major axis and (`gop.pbx`, `gop.pby`,`gop.pbz`) for the semi-minor axis.

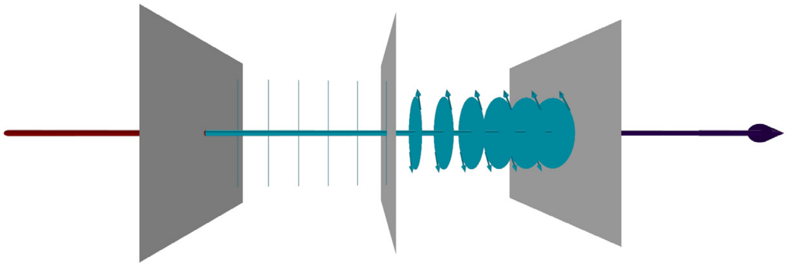


Figure 2-3: Polarization ellipses as a ray crosses a linear polarizer, linear wave retarder, and a second linear polarizer.

Linear Wave Retarder: Application Library path **Ray_Optics_Module/ Tutorials/linear_wave_retarder**

Ray Plot

The **Ray** plot can be added to 1D plot groups. There are two primary ways to use this plot:

1. Plot an expression versus time for all rays, or as a data series operation such as a sum or average over all rays.
2. Plot two expressions against each other, creating a separate curve for each ray, at a specified time.

The following plot is an example of usage case 2. Polychromatic light is transmitted through a multilayer film, and then the intensity at the final time step is plotted against vacuum wavelength for all rays.

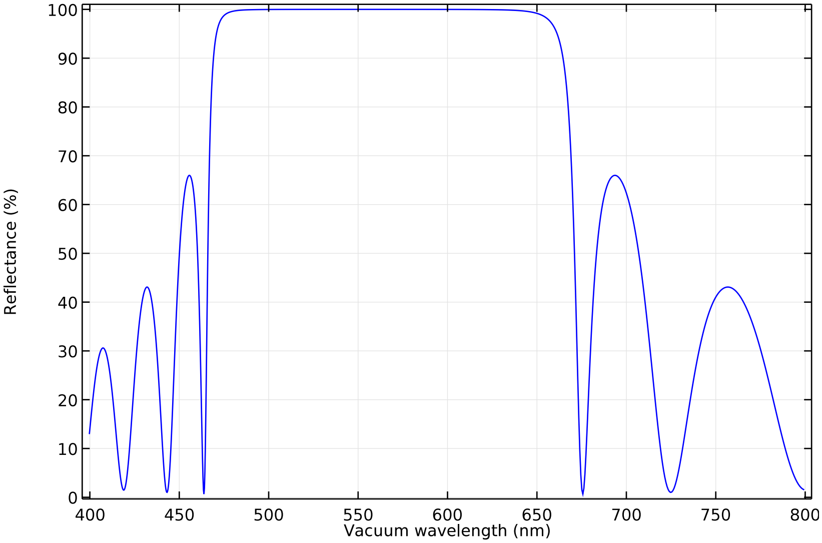


Figure 2-4: Reflectance of a distributed Bragg reflector with 21 dielectric layers is plotted as a function of free-space wavelength.

Distributed Bragg Reflector: Application Library path **Ray_Optics_Module/Prisms_and_Coatings/distributed_bragg_reflector**

Interference Pattern Plot

The pattern of fringes resulting from the interference of two or more rays can be plotted using the dedicated **Interference Pattern** plot. The Interference Pattern plot is available in 2D plot groups and requires a **Cut Plane** data set pointing to a **Ray** data set. The interference pattern is then plotted using the locations and properties of rays as they intersect the cut plane.

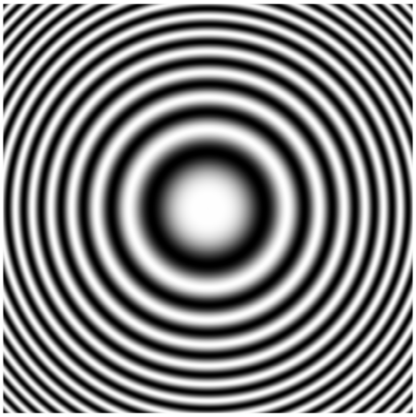


Figure 2-5: Interference pattern resulting from two point sources at slightly different distances from a surface.

The **Interference Pattern** can only be plotted if the ray intensity and phase are solved for; in the physics interface settings, select an appropriate intensity computation method from the **Intensity computation** list, the select the **Compute phase** check box.

For the resulting interference pattern to be physically meaningful, it must be plotted over a region with a length scale that is much smaller than the principal radii of curvature of the incident wavefronts. This is equivalent to the assumption that the wavefront associated with each ray subtends a very small solid angle, and is necessary due to the approximation used to compute the incident intensity.

Interference Pattern Theory

Poincaré Maps and Phase Portraits

A **Poincaré Map** can be used to plot the intersection points of rays with a plane. To use the Poincaré Map, a **Cut Plane** data set must first be defined.

By placing the **Cut Plane** data set at the image plane of an optical system, it is possible to use the **Poincaré Map** to create spot diagrams in order to evaluate the performance of the optical system.

A **Phase Portrait** can be used to plot the positions of rays in an arbitrarily defined phase space. For example, it is possible to plot rays in a 2D space in which one coordinate represents the optical path length and the other coordinate represents intensity. Like the **Poincaré Map**, the **Phase Portrait** can also be used to create spot diagrams.

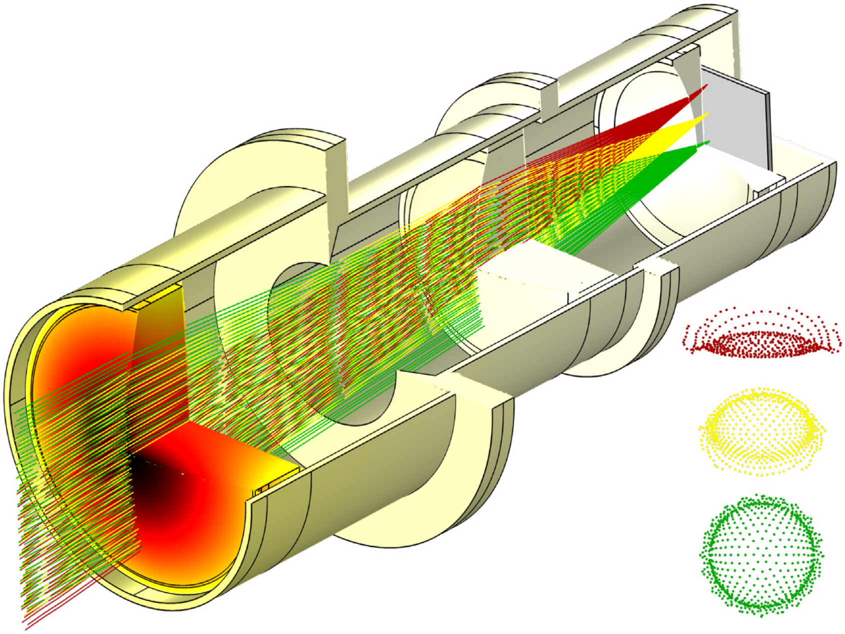


Figure 2-6: Structural-thermal-optical-performance (STOP) analysis of a Petzval lens assembly. The rays are colored according to their initial field angle. Spot diagrams are shown to the right of the main plot.

Petzval Lens STOP Analysis: Application Library path **Ray_Optics_Module/Structural_Thermal_Optical_Performance_Analysis/ petzval_lens_stop_analysis**

Computing Monochromatic Aberrations

The **Optical Aberration** plot and the **Aberration Evaluation** derived values node are used to analyze the performance of lens systems within the limit of the geometrical optics approach. In order to use the **Optical Aberration** plot, the following prerequisites must be met:

- The model component must be 3D.
- An instance of the Geometrical Optics interface must be present and solved for.
- The **Compute optical path length** check box must be selected in the Geometrical Optics **Settings** window before solving.
- An **Intersection Point 3D** data set must be created. This data set must point to a **Ray** data set.
- In the Settings window for the **Intersection Point 3D** data set, **Hemisphere** must be selected from the **Surface type** list. The **Center** is the location of the focus and the **Axis direction** points from the focus toward the center of the exit pupil.

Using the hemisphere defined in the **Intersection Point 3D** data set, a Gaussian reference sphere is defined. The **Optical Aberration** plot or **Aberration Evaluation** feature then computes the optical path length at the intersection points of rays with this Gaussian reference sphere and computes the Zernike coefficients that best fit the optical path difference distribution.

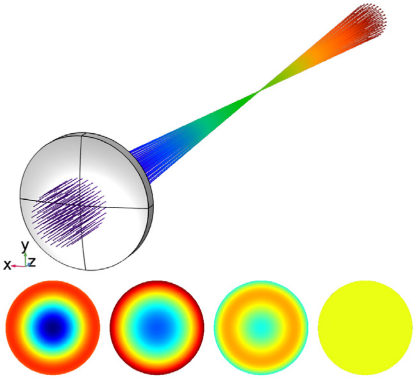


Figure 2-7: Ray trajectories through a simple plano-convex lens (top) and the resulting monochromatic aberrations (bottom). Zernike polynomials: sum of all terms (far left), defocus (left), spherical aberration (right), and all other terms (far right).

For a list of Zernike polynomials and their derivation, see Monochromatic Aberrations and Zernike Polynomials in Theory for the Geometrical Optics Interface.

In the *COMSOL Multiphysics Reference Manual*:

- Interference Pattern
- Optical Aberration
- Aberration Evaluation
- Ray (Plot)
- Ray Trajectories
- Filter for Ray and Ray Trajectories
- Phase Portrait
- Poincaré Map
- Ray (Data Set) and Data Sets
- Ray Evaluation and Derived Values, Evaluation Groups, and Tables
- Plot Groups and Plots

Variables and Component Couplings

This section lists some built-in component couplings and variables for the Geometrical Optics interface.

In this section:

- Ray Statistics
- Global Variables
- Variables for Average Ray Position
- Using Ray Detectors
- Component Couplings

Ray Statistics

The Geometrical Optics interface defines a number of special variables, some of which can only be used during results processing. These variables can be found in the **Ray statistics** section in the **Add/Replace Expression** menus.

In the variable names below, `gop` is the physics interface tag. If multiple instances of the Geometrical Optics interface are different, this tag may differ.

The following variables are defined for each ray:

- Ray index `gop.pidx`. Each ray is assigned a unique index starting from 1 up to the total number of rays. This expression can be used as an input argument to a function. For example, `randomnormal(gop.pidx)` would sample pseudorandom numbers from a normal distribution with unit variance. The random numbers would be distinct for each ray.
- Ray release feature `gop.prf`. If there are multiple release features in a model, it is useful to visualize which rays correspond to each release feature. Rays from each release feature are assigned a unique integer, starting at 1. This variable can also be used to filter ray trajectories in postprocessing so that only the rays released by a specific feature are shown.
- Ray release time `gop.prt`. Usually, all primary rays are released at $t = 0$. To allow other release times to be specified, select the **Allow multiple release times** check box in the physics interface **Advanced Settings** section.

If the **Store ray status data** check box is selected in the physics interface **Additional Variables** section, then the following additional variables are created:

- The release time of a given ray (variable name `gop.rti`). Unlike `gop.prt`, this works for secondary rays and thus allows for extraction of the time at which a secondary ray was released. This includes reflected rays at material discontinuities and higher diffraction orders at gratings.
- Stop time `gop.st`, when a ray gets absorbed at a boundary or annihilated by the Ray Termination feature.
- Final status `gop.fs`. This indicates the status of a ray at a given point in time. When used during postprocessing, the value always indicates the status of the ray at the last time step. The value is an integer which has one of the following values:
  - 0 for unreleased rays.
  - 1 for rays that are still propagating.
  - 2 for frozen rays.
  - 3 for stuck rays.
  - 4 for rays that have disappeared.

Global Variables

The following quantities can be evaluated globally, using the **Global** plot or **Global Evaluation** feature. They could be used in a **Ray Evaluation**, but they would just give the same value for every ray.

- Total number of rays, `gop.Nt`. This total includes both primary and secondary rays, and includes rays that have disappeared or have not been released.
- Total number of rays in selection, `gop.Nsel`. If a selection has been applied to the **Ray** data set, the number of rays in that selection can be evaluated.
- Transmission probability, `gop.alpha` is the ratio of the number of rays in a selection to the total number of rays.

For each ray release feature, the total number of rays released by feature is also defined. This variable has physics feature scope, so for example, a Release from Grid node with tag `relg1` would define a variable called `gop.relg1.Ntf`.

This global variable is uniquely defined for each release feature, and gives the total number of rays that are released by that feature. This includes rays that have disappeared or have otherwise stopped propagating due to interaction with the surrounding boundaries. It does not include any secondary rays.

Variables for Average Ray Position

Some other variables for average ray position, both absolute and relative, are also defined. These variables exist with both physics scope and physics feature scope; for example, the variable `gop.rrms` is the RMS spot size for all rays in the model, but `gop.relg1.rrms` is the RMS spot size only for rays produced by **Release from Grid 1**.

- Components of the average ray position `gop.qavex`, `gop.qavey`, and `gop.qavez`, which are simple arithmetic means over the respective ray coordinates.
- Relative ray position `gop.deltaqx`, `gop.deltaqy`, and `gop.deltaqz`. These are the ray positions relative to the average ray position. for example, the relative x-component `gop.deltaqx` is `qx-gop.qavex`.
- RMS spot size `gop.rrms`.
- Maximum relative radial position `gop.rmaxall`.

Many Ray Optics tutorials use the RMS spot size as a figure-of-merit during postprocessing. For example, see the *Petzval Lens*: Application Library path **Ray_Optics_Module/Lenses_Cameras_and_Telescopes/petzval_lens**

Using Ray Detectors

A Ray Detector is a domain or boundary feature that provides information about rays arriving on a set of selected domains or surfaces from a release feature. The feature provides convenient expressions that can be used in the **Filter** attribute of the **Ray Trajectories** plot, which allows only the rays which reach the ray detector selection to be shown. The following variables are defined by the **Ray Detector** feature, assuming a physics interface with tag `gop` and a detector with tag `rd1`:

- `gop.rd1.Ntf` is number of transmitted rays from the release feature to the ray detector at the end of the simulation.
- `gop.rd1.alpha` is the transmission probability from the release feature to the ray detector.
- `gop.rd1.rL` is a logical expression for ray inclusion. This can be set in the **Filter** node of the **Ray Trajectories** plot in order to visualize the rays which connect the release feature to the detector.

The **Ray Detector** feature defines new variables but does not otherwise affect the solution. Therefore, you can add a **Ray detector** to a completed model and evaluate the variables it defines without recomputing the solution. To do so, right click on the **Study** node and select **Update Solution**. The new variables described above will be immediately available for results processing.

Component Couplings

The purpose of a model is often to compute the sum, average, maximum value, or minimum value of a quantity over a group of rays, such as the average intensity or the maximum path length. An instance of the Geometrical Optics interface with physics interface tag `gop` creates the following component couplings:

- `gop.gopop1(expr)` evaluates the sum of the expression `expr` over the rays. The sum includes all rays that are active, frozen, or stuck to boundaries. It excludes rays that have not yet been released and those that have disappeared.
- `gop.gopop_all1(expr)` evaluates the sum of the expression `expr` over all rays, including rays those that are not yet released or have disappeared. Since the coordinates of unreleased and disappeared rays are not-a-number (`NaN`), the sum may return `NaN` if the model includes unreleased or disappeared rays. An expression such as `gop.gopop1(isnan(qx))` can be used to compute the total number of unreleased and disappeared rays.
- `gop.gopaveop1(expr)` evaluates the average of the expression `expr` over the active, frozen, and stuck rays. Unreleased and disappeared rays contribute to neither the numerator nor the denominator of the arithmetic mean.
- `gop.gopaveop_all1(expr)` evaluates the average of the expression `expr` over all rays. It is likely to return `NaN` if the model includes unreleased or disappeared rays.
- `gop.gopmaxop1(expr)` evaluates the maximum value of the expression `expr` over all active, frozen, and stuck rays.
- `gop.gopmaxop_all1(expr)` evaluates the maximum value of the expression `expr` over all rays.

  The treatment of `NaN` values in maximum component couplings can be platform-dependent, so use caution when evaluating the maximum over all rays including disappeared and unreleased rays.

- `gop.gopminop1(expr)` evaluates the minimum value of the expression `expr` over the active, frozen, and stuck rays.
- `gop.gopminop_all1(expr)` evaluates the minimum value of the expression `expr` over all rays.

  The treatment of `NaN` values in minimum component couplings can be platform-dependent, so use caution when evaluating the minimum over all rays including disappeared and unreleased rays.
- `gop.gopmaxop1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `gop.gopmaxop1(gop.I, qx)` would evaluate the *x*-coordinate `qx` of the ray with the greatest intensity `gop.I`.
- `gop.gopmaxop_all1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` for all rays, including disappeared and unreleased rays.
- `gop.gopminop1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `gop.gopminop1(gop.I, qx)` would evaluate the *x*-coordinate `qx` of the ray with the minimum intensity `gop.I`.
- `gop.gopminop_all1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` for all rays, including disappeared and unreleased rays.

The built-in component couplings defined by the Geometrical Optics interface are summarized in Table 2-1.

TABLE 2-1: BUILT-IN COMPONENT COUPLINGS FOR THE GEOMETRICAL OPTICS INTERFACE

NAME	DESCRIPTION
`gop.gopop1(expr)`	Sum over rays
`gop.gopop_all1(expr)`	Sum over all rays
`gop.gopaveop1(expr)`	Average over rays
`gop.gopaveop_all1(expr)`	Average over all rays
`gop.gopmaxop1(expr)`	Maximum over rays
`gop.gopmaxop_all1(expr)`	Maximum over all rays
`gop.gopminop1(expr)`	Minimum over rays
`gop.gopminop_all1(expr)`	Minimum over all rays
`gop.gopmaxop1(expr, evalExpr)`	Evaluate at maximum over rays

TABLE 2-1: BUILT-IN COMPONENT COUPLINGS FOR THE GEOMETRICAL OPTICS INTERFACE

NAME	DESCRIPTION
`gop.gopmaxop_all1(expr, evalExpr)`	Evaluate at maximum over all rays
`gop.gopminop1(expr, evalExpr)`	Evaluate at minimum over rays
`gop.gopminop_all1(expr, evalExpr)`	Evaluate at minimum over all rays

3

Ray Optics Interfaces

This chapter describes the physics interfaces found under the **Optics>Ray Optics** branch ( ).

In this chapter:

- The Geometrical Optics Interface
- Theory for the Geometrical Optics Interface

The Geometrical Optics Interface

The **Geometrical Optics (gop)** interface ( ), found under the **Optics>Ray Optics** branch ( ) when adding a physics interface, computes the paths of electromagnetic waves in the high frequency limit. The wavefronts are treated as rays that can propagate through optically large geometries and reflect or refract at boundaries. The physics interface includes built-in options to compute the ray intensity, polarization, phase, and optical path length. It is also fully compatible with deformed geometries such as systems with thermal stress.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Medium Properties**, **Material Discontinuity**, and **Ray Properties**. Then, from the **Physics** toolbar, add other nodes that implement, for example, light sources or boundary conditions. You can also right-click **Geometrical Optics** to select physics features from the context menu.

Geometrical Optics Physics Interface Settings

The **Label** is the physics interface name. The default is Geometrical Optics.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `gop`.

RAY RELEASE AND PROPAGATION

The settings in this section affect the way in which primary and secondary rays are released.

Wavelength Distribution of Released Rays

Select an option from the **Wavelength distribution of released rays** list: **Monochromatic** (the default), **Polychromatic, specify vacuum wavelength**, or **Polychromatic, specify frequency**.

- For **Monochromatic** all rays in the model have the same vacuum wavelength and frequency, which is entered as a value or expression in the Ray Properties node.

- For **Polychromatic, specify vacuum wavelength** a degree of freedom is allocated for the vacuum wavelength of each ray in the model. These degrees of freedom are initialized when the rays are released and are controlled by the **Vacuum Wavelength** section in the settings for the ray release features, such as Release from Grid.
- For **Polychromatic, specify frequency** a degree of freedom is allocated for the frequency of each ray in the model. These degrees of freedom are initialized when the rays are released and are controlled by the **Initial Ray Frequency** section in the settings for the ray release features, such as Release from Grid.

Refractive Index of Exterior Domains

Enter a value for the **Refractive index of exterior domains** (dimensionless). The default value is 1. This value of the refractive index is used when tracing rays outside of the domain selection of the Geometrical Optics physics interface. It is also used when rays propagate in the void region outside the geometry. Some limitations apply, as described in the Geometry and Meshing section of the Ray Optics Modeling chapter.

Maximum Number of Secondary Rays

The **Maximum number of secondary rays** prevents an inordinate number of rays from being generated by capping them at the number supplied in the text field. Secondary rays are released when an existing ray is subjected to certain boundary conditions. For example, when a ray undergoes refraction at a Material Discontinuity between different media, the incident ray is refracted and a reflected ray is created; the degrees of freedom for this reflected ray are taken from one of the available secondary rays, which are preallocated when the study begins. Secondary rays are also used to model the interaction of rays with diffraction gratings, using the Grating boundary condition.

If an insufficient number of secondary rays are preallocated, a reflected ray may not be released when an existing ray undergoes refraction, even if some radiation should be reflected at the material discontinuity, and a warning message will be generated by the study. However, if a very large number of secondary rays are preallocated, then the number of degrees of freedom may become unnecessarily large. Thus, the **Maximum number of secondary rays** should only be large enough that all reflected rays which significantly affect the solution can be released. Note that rays undergoing total internal reflection at material discontinuities are not considered secondary rays and do not require extra preallocated degrees of freedom.

Use Geometry Normals for Ray-Boundary Interactions

By default the **Use geometry normals for ray-boundary interactions** check box is cleared. Then, whenever rays are reflected or refracted at boundaries, the surface normal is computed using the underlying mesh discretization of the boundary rather than the exact shape of the geometry itself. If this check box is selected, then the surface normal will instead be computed from an analytic representation of the geometry surfaces, if such an analytic representation can be obtained.

For the simple case of ray reflection by a parabolic edge in 2D, three example plots are shown in Figure 3-1 below. In the leftmost plot, linear geometry shape order has been specified; in other words, **Linear** has been selected from the **Geometry shape order** list in the settings for the model component. The left figure also uses mesh normals. The reflected ray directions are visibly inaccurate because the boundary mesh is very coarse. The center figure uses **Linear** geometry shape order and geometry normals. Although the mesh is equally coarse, the reflected ray directions are much more accurate. The rightmost figure uses **Quadratic** shape order; because the edge is parabolic, this shape order results in reflected ray directions that are exact (to within machine precision) no matter whether mesh normals or geometry normals are used, because quadratic elements can perfectly represent a parabola.

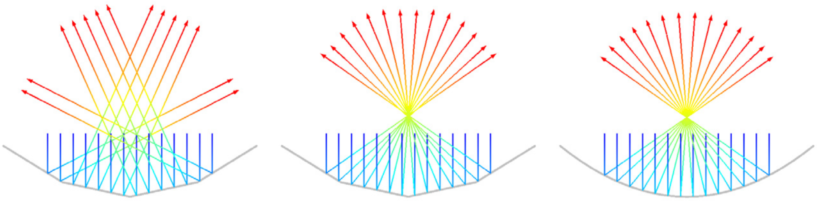


Figure 3-1: Comparison of ray reflections at a parabolic boundary, using various definitions of the mesh normal.

The **Use geometry normals for ray-boundary interactions** check box has no effect on the solution if the mesh can deform. This is true, for example, when the geometry is subjected to structural loads or thermal stresses. In this case the mesh normal is always used.

INTENSITY COMPUTATION

The settings in this section control the treatment of ray intensity and polarization. These settings are also important in multiphysics applications such as ray heating.

Intensity Computation

Select an option from the **Intensity computation** list: **None** (the default), **Compute intensity**, **Compute power**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media**. For **None** no additional variables are computed along the rays. For other options, the ray intensity or ray power is computed as described below.

- For **Compute intensity** auxiliary dependent variables are used to compute the intensity and polarization of each ray. For a complete list of the auxiliary dependent variables that are defined, see Intensity, Wavefront Curvature, and Polarization in Theory for the Geometrical Optics Interface. This option is more accurate and is usually less computationally demanding than **Compute intensity in graded media** but is only valid for computing intensity in homogeneous media. The refractive index can still change discontinuously at boundaries, where the Fresnel equations are automatically used to compute the intensity of the reflected and refracted rays.

  The default name for the intensity variable is `gop.I`.
- For **Compute power** a the total power transmitted by each ray is defined as an auxiliary dependent variable. Information about the ray polarization is also available. The Deposited Ray Power (Boundary) subnode is available for the Wall feature. In addition, if a heat transfer interface such as the Heat Transfer in Solids interface is included in the model, the Ray Heat Source multiphysics node can be used to compute the heat source due to attenuation of rays within domains.

  The default name for the power variable is `gop.Q`.
- The option **Compute intensity and power** functions as a combination of the options **Compute intensity** and **Compute power**.
- The **Compute intensity in graded media** option functions like **Compute intensity** but is valid for both homogeneous and graded-index media. If all media in the model are homogeneous then it is recommended to select **Compute intensity** instead, since it is the more accurate method.
- The **Compute intensity and power in graded media** option functions like **Compute intensity and power** but is valid for both homogeneous and graded-index media. If all media in the model are homogeneous then it is recommended to select **Compute intensity and power** instead, since it is the more accurate method.

Compute Phase

The **Compute phase** check box is only shown if the ray intensity is computed. Select the check box to allocate an auxiliary dependent variable for the phase of each ray. When the phase of each ray is computed, it is possible to plot interference patterns and visualize the instantaneous electric field components of polarized rays in postprocessing. When this check box is selected, the total number of degrees of freedom increases by 1 per ray. This option is based on the assumption that the coherence length of the radiation is arbitrarily large.

Use Corrections for Strongly Absorbing Media

The **Use corrections for strongly absorbing media** check box is shown if the ray intensity is computed. Select the check box to accurately model reflection and refraction of rays at boundaries between strongly absorbing media, in which the imaginary part of the refractive index is very large. This option allocates two or three auxiliary dependent variables per ray based on space dimension. For more information about the way this option affects the intensity calculation, see Refraction in Strongly Absorbing Media in Theory for the Geometrical Optics Interface.

Tolerance for Curvature Tensor Computation

When the **Intensity computation** is set to **Compute intensity in graded media** or **Compute intensity and power in graded media** enter a **Tolerance for curvature tensor computation** (dimensionless). This tolerance is used internally when computing the principal radii of curvature of propagating wavefronts in a graded medium. A larger tolerance makes the solution less accurate but more stable.

ADDITIONAL VARIABLES

The options in this section can be used to solve for additional variables other than those that are used to compute intensity or define the ray frequency. By default, all of the check boxes in this section are cleared so that these variables are not solved for.

Compute Optical Path Length

Select the **Compute optical path length** check box to allocate an auxiliary dependent variable for the optical path length of each ray. The default variable name is `gop.L`. It is possible to reset the optical path length to 0 when rays interact with boundaries.

Count Reflections

Select the **Count reflections** check box to allocate an auxiliary dependent variable for the number of reflections undergone by each ray, including reflections by the Wall and Material Discontinuity features. The default variable name is `gop.Nrefl`. The auxiliary variable begins at 0 when rays are released and is incremented by 1 every time a ray is reflected at a boundary.

Store Ray Status Data

Select the **Store ray status data** check box to add new variables for quantities that cannot necessarily be recovered from the ray trajectory data alone. This is especially true if automatic remeshing is used in a model. The variables created include the following, all of which would be preceded by the physics interface tag (e.g. `gop.`):

- The release time of a given ray (variable name `rti`).
- The time at which a ray stopped at a boundary (variable name `st`).

- The final status of the ray (variable name `fs`). This indicates the status of a ray at the final time step. The value is an integer which has one of the following values:
  - 0 for unreleased rays
  - 1 for rays that are still in the modeling domain
  - 2 for frozen rays
  - 3 for stuck rays
  - 4 for rays that have disappeared

To summarize the total number of rays having each final status, the following global variables are also defined.

TABLE 3-1: GLOBAL VARIABLES BASED ON RAY STATUS

NAME	DESCRIPTION
`fac`	Fraction of active rays at final time step
`fds`	Fraction of disappeared rays at final time step
`ffr`	Fraction of frozen rays at final time step
`fse`	Fraction of secondary rays released
`fst`	Fraction of stuck rays at final time step
`nsr`	Number of released secondary rays
`nsrf`	Number of released secondary rays at final time step
`nsu`	Number of unreleased secondary rays
`nsuf`	Number of unreleased secondary rays at final time step

The global variable names in Table 3-1 all take the unreleased secondary rays into account. For example, suppose an instance of the Geometrical Optics interface includes 100 primary rays and 100 allocated secondary rays. At the last time step, suppose that 80 of the primary rays have disappeared at boundaries and that 40 secondary rays have been emitted, all of which are still active. Then the variable `gop.fac`, the fraction of active rays at the final time step, would have the value $(20 + 40)/(100 + 100)$ or 0.3.

ADVANCED SETTINGS

This section is only shown when **Advanced Physics Options** are enabled (click the **Show** button in the **Model Builder** window). These options are hidden by default because it is only necessary to modify them under very specific circumstances.

Wall Accuracy Order

The **Wall accuracy order** sets the accuracy order of the time stepping used for time steps during which a ray-wall interaction happens. Select an order of **1** to use a forward Euler step and compute the direction or ray propagation both before and after the wall interaction. Select an order of **2** (the default) to use a second-order Taylor method to compute the trajectory before the wall interaction. After the ray-wall interaction a second-order Runge-Kutta method is used. Usually this setting has no effect on the ray tracing calculation because the rays move along straight lines, but the higher wall accuracy order is more accurate if graded-index media are present.

Arguments for Random Number Generation

Select an option from the **Arguments for random number generation** list: **Generate unique arguments** (the default), **Generate random arguments**, or **User defined**. This setting determines how the additional argument to random functions is defined in features such as the Wall boundary condition with the **Diffuse scattering** wall condition. Typically the random numbers are functions of the ray index, position, time, and another argument i, defined as follows:

- For **Generate unique arguments** the additional argument is based on the position of each node in the Model Builder. As a result, random numbers generated in different nodes are created independently of each other, but the same result can be reproduced by running the same study several times.
- For **Generate random arguments** the additional argument is randomly created, causing the random functions to return different results each time the study is run.
- For **User defined** the additional argument is defined by a user input in the **Settings** window for each feature. Independent solutions can be obtained by running a parametric sweep for different values of i.

Allow Multiple Release Times

The **Allow multiple release times** check box, which is cleared by default, allows an array of release times for the rays to be specified in any of the ray release features. If the check box is cleared, all rays are released at time $t = 0$.

Exclude Jacobian Contributions from Ray Equations

By default the **Exclude Jacobian contributions from ray equations** check box is cleared. If this check box is selected, then expressions for the time derivatives of the degrees of freedom on rays are not included in the Jacobian matrix. Excluding contributions to the Jacobian can significantly improve solution time if the number of rays in the model is very large. The drawback is that the Jacobian is not exact, so smaller time steps or path length intervals may be needed to obtain an accurate solution. This drawback is most noticeable when computing the ray intensity or power in an absorbing medium.

Treat Unmeshed Releasing Entities as Errors

By default the **Treat unmeshed releasing entities as errors** check box is selected. Certain ray release features, such as the **Inlet** and **Release** features, compute the ray release positions based on a geometric entity, which must be meshed. While this check box is selected, failure to mesh the selections of such features will result in an error message when running any study in the model. If this check box is cleared, failure to mesh the selections of such features will result in them releasing zero rays, but will otherwise not interrupt the computation. Other physics features will still be able to release rays normally.

Maximum Number of Wall Interactions per Time Step

Enter a value for the **Maximum number of wall interactions per time step**. The default value is 1000. If a ray undergoes more than the specified number of boundary interactions in a single time step taken by the solver, the ray will disappear. This is included as a safeguard to prevent rays from getting stuck in infinite loops if the time between successive ray-wall interactions becomes infinitesimally small.

DEPENDENT VARIABLES

The dependent variables (field variables) are the components of the **Ray position** and **Wave vector.** The name can be changed but the names of fields and dependent variables must be unique within a model. In 3D, the default ray position vector components are `qx`, `qy`, and `qz`; and the default wave vector components are `kx`, `ky`, and `kz`.

- Theory for the Geometrical Optics Interface

List of Geometrical Optics Interface Physics Features

The Geometrical Optics Interface has the following nodes and subnodes (listed in alphabetical order) available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Accumulator (Boundary)
- Accumulator (Domain)
- Auxiliary Dependent Variable
- Axial Symmetry
- Circular Wave Retarder
- Deposited Ray Power (Boundary)
- Deposited Ray Power (Domain)
- Diffraction Order
- Grating
- Ideal Depolarizer
- Illuminated Surface
- Inlet
- Inlet on Axis
- Linear Polarizer
- Linear Wave Retarder
- Material Discontinuity
- Medium Properties
- Mirror
- Mueller Matrix
- Nonlocal Accumulator
- Photometric Data Import
- Ray Continuity
- Ray Detector
- Ray Properties
- Ray Termination
- Release
- Release from Data File
- Release from Edge
- Release from Grid
- Release from Grid on Axis
- Release from Point
- Release from Point on Axis
- Solar Radiation
- Thin Dielectric Film
- Wall

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Medium Properties

Use the **Medium Properties** node to specify the refractive index of the medium. This node is created by default and includes all of the selected domains for the physics interface.

MEDIUM PROPERTIES

Use the settings in this section to specify how the real part of the refractive index is defined. If the ray intensity or power is computed, then you can also specify the imaginary part of the refractive index, which is used to describe absorption within the medium.

Optical Dispersion Models

It is possible to specify the **Refractive index, real part**, n (dimensionless) using one of several optical dispersion models. The options are:

- **Specify refractive index** (the default): use this option to enter a value or expression for the real part of the refractive index n directly. By default the **Refractive index, real part** uses values **From material**. It is also possible to enter a **User defined** value or expression. Only ray propagation in isotropic media can be modeled with the Geometrical Optics interface.
- **Get dispersion model from material**: the dispersion model in each domain is automatically deduced by checking which material properties are defined in each domain. This will often result in different optical dispersion models being used in different domains, especially if the model contains glasses from two or more distinct manufacturers.

  For example, suppose that a model contains two lenses consisting of two different glasses, and that the first glass is defined using **Sellmeier** coefficients whereas the second glass is defined using **Schott** coefficients. Then selecting **Get dispersion model from material** will automatically express the refractive index using the **Sellmeier** equation in the first lens and the **Schott** (polynomial) equation in the second. An alternative way to use different optical dispersion models in different domains is to use more than one instance of the **Medium Properties** node and select the dispersion models manually.

- An optical dispersion model where the refractive index is a function of wavelength may be selected. The following built-in dispersion models are available (See Table 3-3 for details):
    - **Cauchy**,
    - **Conrady**,
    - **Herzberger**,
    - **Schott**,
    - **Schott extended**,
    - **Sellmeier**,
    - **Sellmeier modified, type 1**,
    - **Sellmeier modified, type 2**, or
    - **Temperature-dependent Sellmeier**.

  The coefficients for each of these dispersion models are taken **From material** by default. Alternatively, **User defined** coefficients may be entered.

> ! For the built-in optical dispersion models, the wavelength is always assumed to be in units of microns (μm). For example, in the **Schott** (polynomial) model, the coefficients A_0, A_1, A_2, A_3, etc. have units of 1, μm, μm^2, μm^3, and so on. If another source were to provide these coefficients using nanometers instead of microns, then some manual conversion would be required.

For the options **Specify refractive index** and **Temperature-dependent Sellmeier**, the refractive index is assumed to be absolute, such that $n = 1$ is the refractive index of an ideal vacuum. For all other optical dispersion models, however, the refractive index computed by the dispersion equation is a relative index defined with respect to air at a specified reference temperature and pressure. Thus, for each of these other optical dispersion models, the **Reference temperature** T_{ref} (SI unit: K) and **Reference pressure** P_{ref} (SI unit: Pa) can either be taken **From material** or **User defined**.

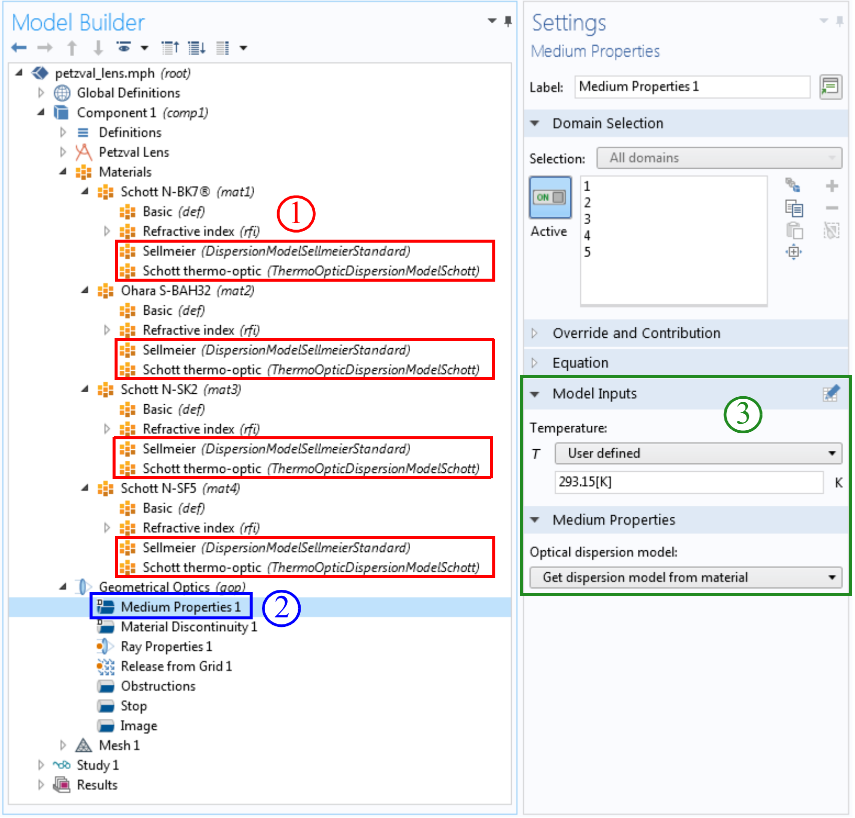
If the reference pressure $P_{\text{ref}} \neq 0$, then it is assumed that the refractive indices are relative to air at the reference temperature and pressure; that is $n = n_{\text{rel}} \times n_{\text{air}}$. The conversion from relative to absolute refractive index is made using a model for the refractive index of air. The **Temperature-dependent Sellmeier** model always returns absolute indices. If **User defined** coefficients are selected, the default coefficients for each of the built-in dispersion models give $n_{\mathrm{rel}} = 1$ (or, $n = 1$, if $P_{\mathrm{ref}} = 0$). See Optical Dispersion Models for further details.

If ray intensity or power is computed, specify the **Refractive index, imaginary part** k (dimensionless). By default the **Refractive index, imaginary part** k uses values **From material**. For **User defined** enter a value or expression. The convention followed by the Geometrical Optics interface is that an the imaginary part of the refractive index is negative in absorbing media. A positive value indicates a gain medium in which the intensity increases as the ray propagates.

How to Automatically Detect Optical Dispersion Models

The option **Get dispersion model from material** is unique because it allows a single **Medium Properties** node to detect different optical dispersion models from a large number of different materials having different domain selections. For example, the materials used in the Petzval Lens tutorial are shown in Figure 3-2 below. To automatically detect the optical dispersion models of the loaded glasses, take the following steps:

1. Add materials to the model using the Optical Material Library. Most of the glasses in this material library use an optical dispersion model, specified by the glass manufacturer, to define the refractive index. As the materials are added, the coefficients used by the optical dispersion model are automatically loaded.
2. Locate the default **Medium Propertie**s node.
3. From the **Optical dispersion model** list, select **Get dispersion model from material**. If the glasses loaded in step 1 also provide thermo-optic coefficients, it is important to specify an accurate value of the **Temperature** as well.



Label: Medium Properties 1

Selection: All domains

Temperature:

T User defined

293.15[K] K

Optical dispersion model:

Get dispersion model from material

Figure 3-2: Workflow for automatically detecting optical dispersion models from the glasses in an optical prescription.

Thermo-Optic Dispersion Models

A temperature-dependent offset in the **Refractive index** may be specified using a thermo-optic dispersion model. Two options are possible:

- **None** (the default): no offset will be applied.
- **Schott**: this is the only built-in thermo-optic dispersion model available. The coefficients used to compute a temperature dependent offset in the refractive index can either be taken **From material** (the default), or **User defined**. A reference temperature is also required. This may also be taken **From material** (the default), or **User defined.**

For further details, see Thermo-Optic Dispersion Models.

Wall

Use the **Wall** node to determine what happens to the rays when contact with a boundary is made.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. If the ray intensity or power is computed, the Thin Dielectric Film subnode is also available. If the ray power is computed, the Deposited Ray Power (Boundary) subnode is also available.

WALL CONDITION

Select a **Wall condition**: **Freeze** (the default), **Specular Reflection**, **Stick**, **Disappear**, **Pass through**, **Diffuse scattering**, **Mixed diffuse and specular reflection**, or **General reflection**.

TABLE 3-2: WALL CONDITION OPTIONS

OPTIONS	DESCRIPTION
Freeze	Select to fix the ray position and wave vector at the instant a wall is struck. So, the ray position no longer changes after contact with the wall and the wave vector of the ray remains at the same value as when the ray struck the wall. This boundary condition is typically used to recover the ray intensity or phase at the instant contact was made with the wall.
Specular Reflection	Select to specularly reflect from the wall, so that the angle of incidence and the angle of reflection are equal.
Stick	Select to fix the ray position at the instant the wall is struck. The wave vector components are set to zero.
Disappear	This option means that the ray is not displayed once it has made contact with the wall. Use it if the ray location after contact with the wall is not of interest.
Pass through	This option allows rays to cross the boundary unimpeded. It does not cause rays to be refracted at the boundary between different materials, though; to model refraction, instead use the Material Discontinuity feature.
Diffuse scattering	Select to reflect rays at a wall according to Lambert's cosine law. That is, the probability of a reflected ray propagating in a given direction within a solid angle $d\omega$ is given by $\cos(\theta)d\omega$ where θ is the angle between the direction of the ray and the wall normal.

TABLE 3-2: WALL CONDITION OPTIONS

OPTIONS	DESCRIPTION
Mixed diffuse and specular reflection	Select to reflect rays at a wall either specularly or according to Lambert's cosine law, based on a user-defined probability.
General reflection	Select to allow an arbitrary direction vector to be specified after a ray makes contact with the wall. The ray direction vector components can be functions of the incident ray direction, phase, or any other quantity.

GENERAL REFLECTION SETTINGS

This section is available when **General reflection** (see Table 3-2) is selected as the **Wall condition**.

Enter values for the **Reflected ray direction vector** $\mathbf{L}_p$ (dimensionless) either in Cartesian coordinates (x, y, z) (the default) or select the **Specify tangential and normal direction vector components** check box to enter coordinates in the tangent-normal coordinate system (t_1, t_2, n). In this case the normal direction is selected so that an incident ray is reflected back into the domain it previously occupied if the specified normal direction vector component is positive. The tangential directions are oriented so that they form a right-handed coordinate system, together with the normal direction.

PRIMARY RAY CONDITION

Use this section to apply two different wall conditions on the same set of boundaries based on a probability or logical expression.

Select a **Primary ray condition**: **None** (the default), **Probability**, or **Expression**. When the default, **None**, is kept, it means that the **Wall condition** is always respected by the incident rays.

Probability

If **Probability** is selected, the **Wall condition** is applied with a certain probability. Enter a value for the **Probability**, γ(dimensionless). If the **Wall condition** is not used, the ray instead behaves according to the Otherwise setting.

The value of γshould always be between 0 and 1.

For example, if the **Wall condition** is set to:

- **Freeze** and γ is set to 0.1, then for every 10 rays that strike the wall, on average one freezes and the remaining 9 rays behave according to the Otherwise setting.
- **Stick** and γ is set to 0.5 then on average half of the rays stick to the wall and the other half behave according to the Otherwise setting.

Expression

If **Expression** is selected, the **Evaluation expression** e (dimensionless) is evaluated whenever a ray strikes the wall. The default expression is 1. If the Evaluation expression is nonzero, the ray behaves according to the **Wall condition**, otherwise the ray behaves according to the Otherwise setting.

Otherwise

The options available for the **Otherwise** setting are the same as for the Wall Condition, except that **General reflection** and **Mixed diffuse and specular reflection** are not available. The **Otherwise** setting can be used to make rays interact with a wall differently with a certain probability or when a certain condition is satisfied. For example, to model reflection at a partially specular surface in which 50% of the ray intensity is absorbed and 30% is reflected specularly:

- Select **Mixed diffuse and specular reflection** as the **Wall condition**,
- set the **Probability of specular reflection** to `0.3/(1-0.5)`,
- set the **Primary ray condition** to **Probability**,
- set the **Probability** to `1-0.5`, and
- select **Freeze** as the **Otherwise** option.

ABSORPTION COEFFICIENT

This section is available when

- the ray intensity or power is solved for in the model, and
- the **Wall condition** is set to any option that can reflect rays, including the following:
  - **Specular Reflection**,
  - **Diffuse scattering**,
  - **General reflection**, or
  - **Mixed diffuse and specular reflection**.

Select one of the following from the **Compute reflected intensity using** list: **Absorption coefficients** or **Reflection coefficients**.

For **Absorption coefficients** enter the **Absorption coefficient** α (dimensionless). The default is 0. The intensity or power of the reflected ray will be proportional to $1 - \alpha$.

For **Reflection coefficients** enter the **Reflection coefficient** r (dimensionless). The default value is 1. The intensity or power of the reflected ray will be proportional to r^2.

If the **Wall condition** is set to **Mixed diffuse and specular reflection**, this section is instead called **Absorption Coefficients** and the absorption coefficients for the diffusely and specularly reflected rays are specified separately. All of the text fields are given subscripts s and d for specularly and diffusely reflected rays, respectively.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

When a ray crosses or touches a boundary, the values of the user-defined auxiliary dependent variables can be changed. The value can be a function of any combinations of ray variables and variables defined on the boundary.

Select the **Assign new value to auxiliary variable** check box or boxes based on the number of auxiliary variables in the model. Then enter the new value or expression in the field. For example, if there is an auxiliary variable, psi, then enter a value for $\mathrm{psi}_{\mathrm{new}}$ in the field. So, to increment the value of psi by 1 when a ray touches or crosses a boundary, enter `psi+1` in the text field for $\mathrm{psi}_{\mathrm{new}}$.

ADVANCED SETTINGS

If the **Primary ray condition** is set to **Probability**, or if the **Diffuse scattering** or **Mixed diffuse and specular reflection** wall condition is used, then the **Wall** feature generates random numbers.

If, in addition, the **Arguments for random number generation** setting is set to **User defined** in the physics interface **Advanced Settings** section, the **Advanced Settings** section is available.

Enter the **Additional input argument to random number generator** i (dimensionless). The default value is `1`.

The **Advanced Settings** section is also shown if the **Compute optical path length** check box is selected under the physics interface **Additional Variables** section. Select the **Reset optical path length** check box to set the optical path length to 0 when a ray touches the wall.

Mirror

Use the **Mirror** node to specularly reflect rays at a boundary. This feature functions as a special case of the **Wall** node, except that the wall condition is always specular reflection.

See Wall for information on the following sections: **Absorption Coefficient**, **New Value of Auxiliary Dependent Variables**, and **Advanced Settings**.

Axial Symmetry

The **Axial Symmetry** node is automatically added to 2D axisymmetric models. The options available for the feature are the same as for Wall, except as follows. The **Boundary Selection** is locked and is only applicable to edges on the symmetry axis. The only wall conditions available are **Specular reflection** (the default), **Freeze**, **Stick**, and **Disappear**. In addition, the **Primary Ray Condition** section is not included. The **Axial Symmetry** boundary condition does not affect ray intensity or polarization, except to introduce a phase delay of π in reflected rays to compensate for the change in the principal curvature direction.

The **Axial Symmetry** condition only applies to straight boundaries that are both on the axis of symmetry and adjacent to a domain in the selection of the physics interface. If rays are propagating in the void region outside of the selected domains, it is possible for the rays to enter the region where $r < 0$, even though no geometry is defined there.

To ensure that rays cannot propagate into the region where $r < 0$, draw line segments on the parts of the symmetry axis where no geometric entities exist, then add the **Wall** node to these edges. Keep in mind that rays can only be subjected to boundary conditions if the corresponding boundaries are meshed.

Accumulator (Boundary)

The **Accumulator** subnode is available from the context menu (right-click the Wall or Material Discontinuity parent node) or from the **Physics** toolbar, **Attributes** menu. Each **Accumulator** subnode defines a variable, called the accumulated variable, on each boundary element in the selection of the parent node. The accumulated variables are incremented when rays interact with the boundaries where they are defined.

ACCUMULATOR SETTINGS

Select an option from the **Accumulator type** list: **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the surface area of the boundary element where it is defined.
- For **Count** the accumulated variable is the sum of the source terms of all rays that hit the boundary element, and is unaffected by the boundary element size.

Select an option from the **Accumulate over** list: **Ray-wall interactions** (the default) or **Rays in boundary elements**.

- For **Ray-wall interactions** the accumulated variable is affected by all rays that hit the boundary element.
- For **Rays in boundary elements** the accumulated variable is only affected by rays that freeze or stick to the boundary element.

Enter the **Accumulated variable name**. The default is `rpb`. The accumulated variable is defined as `<scope>.<name>`, where `<scope>` includes the name of the physics interface node, parent boundary condition, and the **Accumulator** node, and `<name>` is the accumulated variable name.

For example, if the **Accumulator** subnode is added to a Wall node in an instance of the Geometrical Optics interface using the default variable name `rpb`, the accumulated variable name might be `gop.wall1.bacc1.rpb`.

Enter a **Source** R. The unit of the source term depends on the settings in the **Units** section. Whenever a ray collides with a boundary element in the selection of the parent node, the accumulated variable in that element is incremented by the source term. If the **Accumulator type** is set to **Density**, the source term is divided by the area of the boundary element (in 3D) or the length of the boundary element (in 2D).

For example, if the source is `1` and **Count** is selected from the **Accumulator type** list, a variable is created to store the number of times each boundary element is hit by a ray.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

The accumulated variables are computed using discontinuous shape functions. Select the **Compute smoothed accumulated variable** check box to compute a smoothed accumulated variable by computing the average value of the variable within a sphere of a user-defined radius. Then enter a **Smoothing radius** r (SI unit: m). The default is 1 m.

Accumulator Theory: Boundaries

Material Discontinuity

The **Material Discontinuity** node is the default feature on all boundaries.

The Geometrical Optics interface always applies reflection and refraction at boundaries between different media using a deterministic ray splitting approach. The direction of the refracted ray is computed using Snell's law, based on the refractive index on either side. If extra degrees of freedom have been allocated for secondary rays, a reflected ray is also released. If the incident ray undergoes total internal reflection, no refracted ray is produced and no secondary rays are needed to release the reflected ray.

If the ray intensity or power is solved for in the model, the **Material Discontinuity** feature computes the new values of these variables for the reflected and refracted rays.

The maximum number of reflected rays can be controlled via the **Maximum number of secondary rays** text field, which is found under the **Ray Release and Propagation** section for the physics interface.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. The Thin Dielectric Film subnode is also available if the following conditions are met:

- The ray intensity or power is being computed, and
- One of the following options is selected from the **Thin dielectric films on boundary** list in the **Coatings** section: **Add layers to surface** or **Add layers to surface, repeating**.

COATINGS

This section is available if ray intensity or power is being computed in the model. Use the options in this section to add thin dielectric layers to the boundary between the media. These thin dielectric layers are used, together with the refractive indices on either side, to modify the Fresnel equations for the reflection and transmission coefficients at the boundary.

The thicknesses of the thin dielectric layers must be small relative to the coherence length of the radiation. They are usually comparable in length scale to the free-space wavelength.

Select an option from the **Thin dielectric films on boundary** list. The default is **None**. The following options are available.

- **None**: no dielectric films are on the boundary. The intensity of the reflected and refracted light is computed solely using the Fresnel equations with the refractive indices of the two adjacent domains.
- **Add layers to surface**: you can add any number of thin dielectric films to the boundary by adding one or more Thin Dielectric Film subnodes; right click the **Material Discontinuity** node or select **Thin Dielectric Film** from the **Physics** toolbar, **Attributes** menu. If multiple thin films are added to a single surface, they are arranged in the same order as the corresponding **Thin Dielectric Film** subnodes in the Model Builder, from the upside of the boundary toward the downside.

Anti-Reflective Coating with Multiple Layers: Application Library path **Ray_Optics_Module/Prisms_and_Coatings/antireflective_coating_multilayer**

- **Add layers to surface, repeating**: this option functions like the **Add layers to surface** option described above. Add individual dielectric coatings to the boundary using the **Thin Dielectric Film** subnode. In addition, enter a value or expression for the **Number of repeating unit cells** N (dimensionless). The default value is 3. In the settings windows for the **Thin Dielectric Film** subnodes, you can decide which layers constitute a unit cell that will be repeated the specified number of times. Use this option to specify periodic arrangements with a large number of repeatind dielectric layers, without having to add each layer manually.

Distributed Bragg Reflector: Application Library path **Ray_Optics_Module/Prisms_and_Coatings/distributed_bragg_reflector**

- **Anti-reflective coating**: the incident ray is refracted with unit transmittance and zero reflectance.
- **Single-layer coating, specified thickness**: this is a simplified version of the **Add layers to surface** option that only allows a single dielectric layer. Enter a value or expression for the **Film refractive index** n (dimensionless). The default value is 1. Then enter a value or expression for the **Film thickness** t (SI unit: m). The default value is 1 µm.

- **Specify reflectance**: enter a value or expression for the reflectance at the boundary directly. Enter a value or expression for the **Reflectance** R (dimensionless). The default is 0.1. If the **Specify different values for s- and p-polarization** check box is selected, you can instead enter separate values for the reflectance of p- and s-polarized light, called R_p and R_s respectively. The layers are considered non-absorbing so that the sum of the reflectance and transmittance is unity.
- **Specify transmittance**: enter a value or expression for the transmittance at the boundary directly. Enter a value or expression for the **Transmittance** T (dimensionless). The default is 0.9. If the **Specify different values for s- and p-polarization** check box is selected, you can instead enter separate values for the transmittance of p- and s-polarized light, called T_p and T_s respectively. The layers are considered non-absorbing so that the sum of the reflectance and transmittance is unity.

Real and Ideal Optical Coatings

The options **Anti-reflective coating**, **Specify reflectance**, and **Specify transmittance** all include a check box called **Treat as single layer dielectric film**, which is cleared by default. While this check box is cleared, the specified reflectance or transmittance applies to all rays, regardless of wavelength or angle of incidence, and the remaining inputs in this section are not shown. This makes complicated multilayer films easier to set up because you can enter the reflectance or transmittance explicitly without having to know the properties of each layer in the coating. If the coating properties depend on the wavelength or angle of incidence, you can usually build this dependence into the model by defining the film reflectance or transmittance as an **Interpolation function** of the ray properties. The tradeoff is that accurate information about the discontinuities in ray phase is lost, even if the **Compute phase** check box is selected in the physics interface **Intensity Computation** section, because this information depends on the complex-valued Fresnel coefficients.

If the **Treat as single layer dielectric film** check box is selected, then instead the **Material Discontinuity** automatically computes the thickness of a single layer needed to produce the given reflectance or transmittance. The drawback is that the specified reflectance or transmittance will only be attained for rays at a specific wavelength and angle of incidence.

For **Specify reflectance** and **Specify transmittance** enter the **Film refractive index** n (dimensionless). The default is 1. If it is impossible to construct a single layer of this refractive index and obtain the specified reflectance or transmittance, then the boundary is treated as having no dielectric films. For **Anti-reflective coating**, the refractive index of the coating is not an input because it is always the geometric mean of the refractive indices in the two adjacent domains.

For **Anti-reflective coating**, **Specify reflectance**, and **Specify transmittance** enter the following:

- **Vacuum wavelength for specified film properties** λ_0 (SI unit: m). The default is 660 nm.
- **Angle of incidence for specified film properties** θ_i (SI unit: m). The default is 0.
- Select an option from the **Angle of incidence specification** list: **With respect to upside** (the default) or **With respect to downside**. You can see which side is the upside by selecting the **Show boundary normal** check box (see **Advanced Settings** below); the normal vector rendered in the Graphics window points from the upside to the downside.
- Select an option from the **Specified film behavior applies to** list: **S-polarized radiation** (the default) or **P-polarized radiation**.

RAYS TO RELEASE

Use the inputs in this section to control whether the incident ray is split into a reflected and refracted ray at the material discontinuity. Select an option from the **Release reflected rays** list: **Always** (the default), **Never**, or **Based on logical expression**.

- If **Always** is selected, whenever a ray hits the material discontinuity, the incident ray is refracted across the boundary, and a reflected ray is produces using some of the preallocated degrees of freedom for secondary rays.
- If **Never** is selected, the incident ray is refracted, but no reflected ray is produced. This does not prevent a ray from undergoing total internal reflection at the boundary.
- If **Based on logical expression** is selected, enter a value or expression for the **Evaluation expression** e (dimensionless). The default is 1. A reflected ray is produced only if the value of this expression is nonzero for the incident ray.

If the ray intensity is solved for in the model, enter a **Threshold intensity** I_{th} (SI unit: W/m^2). The default is 1 mW/m^2. If the interaction of a ray with a material discontinuity would create a reflected ray of intensity less than the threshold intensity, the release of this reflected ray is suppressed.

If the ray power is solved for in the model, enter a **Threshold power** Q_{th} (SI unit: W). The default is 10^{-3} mW. If the interaction of a ray with a material discontinuity would create a reflected ray of power less than the threshold power, the release of this reflected ray is suppressed.

If both the **Evaluation expression** and the **Threshold intensity** are specified, a reflected ray is only released if the value of the expression is nonzero and the intensity of the reflected ray exceeds the threshold. In other words, a ray must satisfy all of the criteria in the **Rays to Release** section in order to be released, not just one of them.

The release of reflected rays at a material discontinuity can be suppressed to prevent an arbitrarily large number of degrees of freedom from being used to model rays of extremely low intensity. Keep in mind that this can lead to a reduction in the total energy of the system.

AUXILIARY DEPENDENT VARIABLES, REFRACTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the refracted ray. These settings are the same as in the New Value of Auxiliary Dependent Variables section for the Wall feature.

AUXILIARY DEPENDENT VARIABLES, REFLECTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the reflected ray. These settings are the same as in the New Value of Auxiliary Dependent Variables section for the Wall feature.

ADVANCED SETTINGS

Select the **Show boundary normal** check box to view the boundary normal in the **Graphics** window.

> ! Visualization of the boundary normal is important when adding multiple Thin Dielectric Film subnodes, since the thin films are oriented from the upside of the boundary to the downside in the same order as their corresponding nodes in the Model Builder.

If the **Compute optical path length** check box is selected in the physics interface **Additional Variables** section, select the **Reset optical path length** check box to set the optical path length of reflected and refracted rays to 0. Otherwise both the reflected and refracted ray inherit the optical path length of the incident ray.

Material Discontinuity Theory

Thin Dielectric Film

Use the **Thin Dielectric Film** subnode to model the interaction of rays with walls or material discontinuities that are coated with thin, transparent dielectric layers. The presence of these layers modifies the Fresnel coefficients used to compute the intensity or power of the reflected and refracted rays. The films must be very thin relative to the coherence length of the radiation and are usually comparable to or smaller than the wavelength. The subnode is available from the context menu (right-click the Material Discontinuity or Wall parent node) or from the **Physics** toolbar, **Attributes** menu.

The **Thin Dielectric Film** subnode is applied to all boundaries in the selection of the **Wall** or **Material Discontinuity** parent node. The effects of the **Thin Dielectric Film** subnodes are applied when computing the coefficients of reflection and transmission, which are necessary for the reinitialization of Stokes parameters during reflection and refraction if the intensity is solved for in the model.

FILM PROPERTIES

Enter a **Film refractive index** n (dimensionless). The default value is 1. Then enter a **Film thickness** t (SI unit: m). The default is 1 µm.

REPEATING MULTILAYER FILMS

If **Add layers to surface, repeating** is selected from the **Thin dielectric films on boundary** list in the **Coatings** section in the settings window for the parent node, this section is used to determine which layers are part of the repeating unit cell of a periodic multilayer film. If the **Repeat layer in multilayer films** check box is selected, the layer is repeated a number of times equal to the **Number of repeating unit cells** specified in the parent node settings window. If the check box is cleared, the layer will appear only once in multilayer films.

Thin Dielectric Film Theory

Ray Properties

The **Ray Properties** node is added by default. Only one instance of this node can exist for each physics interface. The **Ray Properties** node defines the equations for ray propagation and may also include inputs for some ray properties.

RAY PROPERTIES

This section is shown when **Monochromatic** is selected from the **Wavelength distribution of released rays** list in the physics interface **Ray Release and Propagation** section. Otherwise the frequency or vacuum wavelength of released rays is controlled separately by each ray release feature.

Select an option from the **Ray property specification** list: **Specify vacuum wavelength** (the default) or **Specify frequency**.

- For **Specify vacuum wavelength** enter a **Vacuum wavelength** λ_0 (SI unit: m). The default is 660 nm.
- For **Specify frequency** enter a **Ray frequency** v (SI unit: Hz). The default is 4.54×10^{14} Hz.

Photometric Data Import

The **Photometric Data Import** node is only available in 3D, and only if the ray intensity is solved for in the model.

Use the **Photometric Data Import** node to import photometric data files that can be used to initialize the ray intensity and power as a function of the initial ray direction. The **Photometric Data Import** node supports the `*.ies` file extension, the standard photometric data file format of the Illuminating Engineering Society of North America (IESNA).

PHOTOMETRIC DATA IMPORT

Enter a **File name**, or click the **Browse** button to locate a photometric data file. When a valid file path has been entered, click the **Import** button to load the photometric data into the model. A shortened version of the name of the photometric data file will then be displayed, along with a **Discard** button. Click the **Discard** button to discard the loaded photometric data and clear the **File name** field.

After creating an instance of the **Photometric Data Import** feature and loading an appropriately formatted IES file into the model, initialize the ray intensity as a function of initial ray direction by changing the following settings in a ray release feature such as the Release or Release from Grid feature:

- Select **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** from the **Ray direction vector** list in the physics feature **Ray Direction Vector** section.
- Locate the **Total Source Power** section. From the **Intensity initialization** list, select an instance of the **Photometric Data Import** feature.
- Enter values or expressions for the **Photometric horizontal** and **Photometric zero** directions.

Photometric Data Formatting

The format of the imported photometric data file must be consistent with ANSI/IESNA LM-63-02 (R2008), *IESNA Standard file format for the electronic transfer of photometric data and related information*, Illuminating Engineering Society (2002).

Release

Use the **Release** node to release rays within domains based on arbitrary expressions or based on the positions of the mesh elements.

RELEASE TIMES

This section is only available when the **Allow multiple release times** check box has been selected in the physics interface **Advanced Settings** section. Enter **Release times** (SI unit: s) or click the **Range** button (⎵) to select and define a range of specific times. At each release time, rays are released with initial position and ray direction vector as defined next.

INITIAL POSITION

Select an **Initial position**: **Density** (the default) or **Mesh based**.

Density

For **Density** enter a value for the **Number of rays per release** N (dimensionless). The default is 100. Then enter a value or expression for the **Density proportional to** ρ (dimensionless). The default is 1.

The **Density proportional to** ρ can be an expression rather than a number; the resulting ray distribution approximately has a number density that is proportional to this expression. The resulting distribution looks a bit random, and it depends on the order in which the mesh elements are numbered. The distribution is probably not exactly the same in different COMSOL Multiphysics versions, but the total number of rays released is always N.

! The **Density proportional to** expression must be strictly positive.

Advanced Options for Density-Based Release

The following options can be adjusted to make the number density of rays more closely conform to the **Density proportional to** expression.

Select a **Release distribution accuracy order** between **1** and **5** (the default is **5**), which determines the integration order that is used when computing the number of rays to release within each mesh element. The higher the accuracy order, the more accurately rays will be distributed among the mesh elements.

The **Position refinement factor** (default 0) must be a nonnegative integer. When the refinement factor is 0, each ray is always assigned a unique position, but the density is taken as a uniform value over each mesh element. If the refinement factor is a positive integer, the distribution of rays within each mesh element is weighted according to the density, but it is possible for some rays to occupy the same initial position. Further increasing the **Position refinement factor** increases the number of evaluation points within each mesh element to reduce the probability of multiple rays occupying the same initial position.

Mesh Based

For **Mesh based** the rays are released from a set of positions determined by a selection of geometric entities (of arbitrary dimension) in the mesh. Given a **Refinement factor** between 1 and 5, the centers of the refined mesh elements are used. Thus, the number of positions per mesh element is `refine^dim`, except for pyramids, where it is `(4*refine`^{2}`-1)*refine/3`.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression** a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension.
- For **Spherical** a number of rays are released at each point, sampled from a spherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50.
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis** $\mathbf{r}$ based on space dimension.
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis** $\mathbf{r}$ based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians.
- The **Lambertian** option is only available in 3D. A number of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis** $\mathbf{r}$ based on space dimension.

If **Conical** is selected, select an option from the **Conical distribution** list: **Uniform density** (the default), **Specify polar and azimuthal distributions**, **Marginal rays only**, or **Marginal and axial rays only**.

- For **Uniform density** rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.
- For **Specify polar and azimuthal distributions** specify the **Number of polar angles** N_ϕ (dimensionless) and the **Number of azimuthal** angles N_θ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray ($\phi = 0$) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\phi \times N_\theta + 1$.

- For **Marginal rays only** the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- For **Marginal and axial rays only** the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

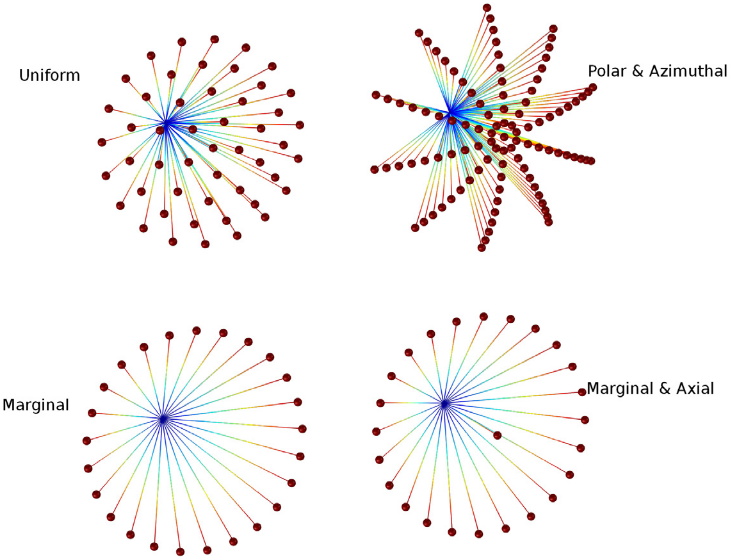


Figure 3-3: Comparison of the four available cone-based release distributions.

For **Spherical**, **Hemispherical**, **Conical**, and **Lambertian**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using an algorithm that seeks to distribute the rays as evenly as possible in wave vector space. This algorithm will give the same initial ray directions whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudorandom numbers. The result may be the same when rerunning the study multiple times on the same computer, but the solution is likely to be different on different architectures.

INITIAL RAY FREQUENCY

This section is available when **Polychromatic, specify frequency** is selected from the **Wavelength distribution of released rays** list in the physics interface **Ray Release and Propagation** section.

Select a **Distribution function**: **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of values**.

When **None** is selected, enter an initial value ν_0 (SI unit: Hz). The default value is 4.54×10^{14} Hz.

Select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any of these selections, the **Number of values** sets the number of points in the distribution function. Enter a user-defined **Mean** (default 4.54×10^{14} Hz) and **Standard deviation** (default 10^{14} Hz). Select **List of values** to enter a list of distinct frequency values.

VACUUM WAVELENGTH

This section is available when **Polychromatic, specify vacuum wavelength** is selected from the **Wavelength distribution of released rays** list in the physics interface **Ray Release and Propagation** section.

Select a **Distribution function**: **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of values**.

When **None** is selected, enter a value or expression for the **Vacuum wavelength** λ_0 (SI unit: m). The default is 660 nm. All rays released by this feature will have the same wavelength.

Select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any of these selections, the **Number of values** sets the number of points in the distribution function. Enter a user-defined **Mean** (default 660 nm) and **Standard deviation** (default 100 nm). Select **List of values** to enter a list of distinct wavelength values directly.

INITIAL PHASE

This section is available when the **Compute phase** check box is selected under the physics interface **Intensity Computation** section. Enter an **Initial phase** Ψ_0 (SI unit: rad). The default value is 0.

INITIAL INTENSITY

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Enter a value for the **Initial intensity** I_0 (SI unit: $\mathrm{W/m^2}$). The default is 1000 $\mathrm{W/m^2}$.

INITIAL RADII OF CURVATURE

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Select a **Wavefront shape**. In 3D the available options are **Plane wave** (the default), **Spherical wave**, and **Ellipsoid**. In 2D the available options are **Plane wave** (the default) and **Cylindrical wave**.

- For an idealized plane wave the radii of curvature would be infinite. However, because the algorithm used to compute intensity requires finite values, when **Plane wave** is selected the initial radii of curvature are instead given an initial value that is 10^8 times greater than the characteristic size of the geometry.
- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid**, enter the **Initial radius of curvature, 1** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, 1** $\mathbf{e}_{1,0}$ (dimensionless).

Principal Radii of Curvature

INITIAL POLARIZATION

This section is available when the ray intensity or power is solved for.

Select an **Initial polarization type**: **Unpolarized** (the default), **Fully polarized**, or **Partially polarized**.

Select an **Initial polarization**: **Along principal curvature direction** (the default) or **User defined**.

- For **Fully polarized** and **Partially polarized** rays in 3D enter an **Initial polarization parallel to reference direction** $a_{1,0}$ (dimensionless), **Initial polarization perpendicular to reference direction** $a_{2,0}$ (dimensionless), and **Initial phase difference** δ_0 (SI unit: rad).
- For **Fully polarized** and **Partially polarized** rays in 2D enter an **Initial polarization, in plane** $a_{xy,1}$ (dimensionless), **Initial polarization, out of plane** $a_{z,0}$ (dimensionless), and **Initial phase difference** δ_0 (SI unit: rad).
- For **User defined** also enter an **Initial polarization reference direction** $\mathbf{u}$ (dimensionless).

For **Partially polarized**, also enter an **Initial degree of polarization** P_0 (dimensionless).

TOTAL SOURCE POWER

This section is available:

- when the ray intensity is solved for, and
- when **Spherical**, **Hemispherical**, or **Conical** is selected as the **Ray direction vector**.
- It is also available when the ray power is solved for, and then any choice of **Ray direction vector** displays this section.

Select an option from the **Intensity initialization** list. By default **Uniform distribution** is selected. If any Photometric Data Import nodes have been added to the model then they can also be selected from the list.

If **Uniform distribution** is selected, enter a **Total source power** P_{src} (SI unit: W). The default is 1 W. In 2D, instead enter the **Total source power per unit thickness** P_{src} (SI unit: W/m). The default is 1 W/m. If any **Photometric Data Import** feature is selected from the list, the source power is instead obtained directly from the imported photometric data (IES) file. Enter values or expressions for the components of the **Photometric horizontal** $\mathbf{p}_h$ (dimensionless) and **Photometric zero** $\mathbf{p}_z$ (dimensionless). By default these vectors point in the positive x- and z-axes, respectively.

The relationship between the photometric horizontal, photometric vertical, and ray direction vector, and its effect on the initial ray intensity, is explained in ANSI/IESNA LM-63-02 (R2008), *IESNA Standard file format for the electronic transfer of photometric data and related information*, Illuminating Engineering Society (2002).

Currently the **Photometric Data Import** feature does not support the options `TILT=INCLUDE` or `TILT=<FILENAME>` that are included in some IES files. Only `TILT=NONE` is allowed.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the **Auxiliary Dependent Variable** nodes added to the model, select a **Distribution function** for the initial value of the auxiliary dependent variables and whether the initial value of the auxiliary dependent variables should be a scalar value or sampled from a distribution function.

The number of rays simulated can increase substantially and the following options are available for each **Auxiliary Dependent Variable** added to the model.

When **None** is selected, enter an initial value. The symbol for the initial value is the auxiliary variable name followed by a subscript 0, so for the default name rp the initial value has symbol rp_0.

For the initial value of the auxiliary dependent variables, select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any selection, the **Number of values** sets the number of points in the distribution function. Enter a user-defined **Mean** (default 0) and **Standard deviation** (default 1). Select **List of values** to enter a set of numerical values directly.

By default auxiliary dependent variables are initialized after all other degrees of freedom. Select the **Initialize before wave vector** check box to compute the initial value of the auxiliary dependent variable immediately after computing the initial wave vectors of the rays. By selecting this check box it is possible to define the initial ray direction as a function of the auxiliary dependent variables.

Deposited Ray Power (Domain)

The **Deposited Ray Power** node is used to compute a heat source based on the attenuation of rays as they propagate through a domain with a complex refractive index. This feature is available when the ray power is solved for. The **Deposited Ray Power** node defines a dependent variable with default name `<name>.Qsrc,` where `<name>` is the name of the Geometrical Optics physics interface, for example `gop.Qsrc`.

Attenuation Within Domains

Deposited Ray Power (Boundary)

When the ray power is solved for, the **Deposited Ray Power** subnode is available from the context menu (right-click the Wall, parent node) or from the **Physics** toolbar, **Attributes** menu. The **Deposited Ray Power** subnode computes the total incident energy flux on a surface based on the power of incident rays.

SMOOTHING

Select the **Compute smoothed accumulated variable** check box to enter a **Smoothing radius** r (SI unit: m) The smoothing radius functions like the corresponding setting for the Accumulator (Boundary) node.

Accumulator (Domain)

Use the **Accumulator** node to define additional degrees of freedom on a domain. Each **Accumulator** defines a variable, called the accumulated variable, on each domain element in the selection list. The values of the accumulated variables are determined by the properties of rays in each domain element.

ACCUMULATOR SETTINGS

Select an **Accumulator type**: **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count** the accumulated variable is unaffected by the element size.

Select an option from the **Accumulate over** list: **Elements** (the default) or **Elements and time**.

- For **Elements** the value of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume.
- For **Elements and time** the time derivative of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume. As each ray propagates through a series of mesh elements, it leaves behind a contribution to the accumulated variable that remains even after the ray has moved on.

Enter the **Accumulated variable name**. The default is `rpd`. The accumulated variable is defined as `<name>.<varname>`, where `<name>` is the physics interface name and `<varname>` is the accumulated variable name. For example, in an instance of the Geometrical Optics interface with default name `gop`and default accumulated variable name `rpd`, the variable would be named `gop.rpd`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

If **Elements and time** is selected from the **Accumulate over** list, select an option from the **Source interpolation** list: **Constant**, **Linear** (the default), **Quadratic**, or **Exponential**. The **Source interpolation** determines what functional form the **Source** is assumed to follow during each time step taken by the solver. This information is used to compute the accumulated variable in mesh elements that the rays pass through during each time step.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

Nonlocal Accumulator

Use the **Nonlocal Accumulator** subnode to communicate information from a ray's current position to the position from which it was released.

The subnode is available from the context menu (right-click the Inlet parent node) or from the **Physics** toolbar, **Attributes** menu.

Each **Nonlocal accumulator** subnode defines a variable, called the accumulated variable, that is computed using variables defined on rays released by the parent **Inlet** node or on domains and boundaries encountered by such rays.

ACCUMULATOR SETTINGS

Select an **Accumulator type**: **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count** the accumulated variable is unaffected by the element size.

Select an option from the Accumulate over list: **Elements** (default) or **Elements and time**.

- For **Elements** the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays.
- For **Elements and time** the time derivative of the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays, and thus the accumulated variable considers the time history of rays in the modeling domain instead of just their current values.

Enter the **Accumulated variable name**. The default is `rpi`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

Select a **Source geometric entity level**: **Domains**, **Boundaries**, or **Domains and boundaries**.

- If **Domains** is selected, rays only contribute to the accumulated variable on their releasing surface if they are still active; that is, they are still propagating through a domain.
- If **Boundaries** is selected, the rays only contribute to the accumulated variable if they have become stuck or frozen to a boundary somewhere in the model.
- If **Domains and boundaries** is selected, all of the active, stuck, and frozen rays released by a feature can contribute to the accumulated variable.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

Select the **Compute smoothed accumulated variable** check box to enter a **Smoothing radius** r (SI unit: m). The default is 0.1 m.

Inlet

Use the **Inlet** node to determine how to release rays on an interior or exterior boundary in a specific direction.

The Nonlocal Accumulator subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

See Release for information on the following sections: **Release Times**, **Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Total Source Power**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

COORDINATE SYSTEM SELECTION

It is possible to specify the initial ray direction in terms of the global coordinates or in another coordinate system defined for the model Component. Select an option from the **Coordinate system** list. By default **Global coordinate system** is selected. If other coordinate systems are defined, they can also be selected from the list. When specifying the initial ray direction (see the **Ray Direction Vector** section), direction components can be specified using the basis vectors of whichever coordinate system has been selected from the list.

When a coordinate system other than **Global coordinate system** is selected from the **Coordinate system** list, arrows will appear in the Graphics window to indicate the orientation of the basis vectors of the coordinate system on the selected boundaries.

INITIAL POSITION

Select an **Initial position**: **Uniform distribution** (the default for 2D components) **Projected plane grid** (the default for 3D components), **Mesh based**, or **Density**. Mesh based and Density have the same settings as described for the Release node.

For 2D components, if **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

For 3D components, if **Projected plane grid** is selected, enter the **Number of rays per release** N (dimensionless). The rays are distributed on a plane grid in planes that are approximately tangential to the selected boundaries (for a plane boundary, you get a uniform distribution).

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression** a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the initial direction using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal**

vector components check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).

- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is π/3 radians. Select the **Specify tangential and normal vector components** check box to specify the cone axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- The **Lambertian** option is only available in 3D. A number of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n)

When the **Specify tangential and normal vector components** check box is selected, arrows indicating the normal direction on the selected boundaries will appear in the Graphics window.

Note that the normal direction may be opposite the built-in variable for the boundary normal (for example, `nx`, `ny`, and `nz`) to ensure that a positive value causes rays to be released into the simulation domain. This often occurs when the Inlet is applied to exterior boundaries.

When the normal direction used by the Inlet feature is opposite the normal vector defined by the geometry, the tangential directions are similarly inverted to ensure that the boundary coordinate system is right-handed.

If **Conical** is selected, select an option from the **Conical distribution** list: **Uniform density** (the default), **Specify polar and azimuthal distributions**, **Marginal rays only**, or **Marginal and axial rays only**.

- For **Uniform density** rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.

- For **Specify polar and azimuthal distributions** specify the **Number of polar angles** N_ϕ (dimensionless) and the **Number of azimuthal** angles N_θ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray ($\phi = 0$) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\phi \times N_\theta + 1$.
- For **Marginal rays only** the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- For **Marginal and axial rays only** the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

The four **Conical distribution** options are illustrated in Figure 3-3.

For **Hemispherical**, **Conical**, and **Lambertian**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudo-random numbers.

INITIAL RADII OF CURVATURE

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Select a **Wavefront shape**. In 3D the available options are **From surface curvature** (the default), **Plane wave**, **Spherical wave**, and **Ellipsoid**. In 2D the available options are **Plane wave** (the default) and **Cylindrical wave**.

- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid** (3D only), enter the **Initial radius of curvature, 1** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, 1** $\mathbf{e}_{1,0}$ (dimensionless).

Principal Radii of Curvature

Inlet on Axis

Use the **Inlet on Axis** node to release rays from specified edges along the axis of symmetry in a 2D axisymmetric geometry. It functions like the Inlet node, except that it can only be applied to the axis of symmetry and that the options in the **Ray Direction Vector** section are slightly different.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position**: **Uniform distribution** (the default), **Mesh based**, or **Density**. **Mesh based** and **Density** have the same settings as described for the Release node.

If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The default is 100. The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Spherical**, or **Conical**.

- For **Expression** a single ray is released in the specified direction. Enter the radial and axial components of the **Ray direction vector** $\mathbf{L}_0$ (dimensionless).
- For **Spherical** a number of rays are released at each point, from the negative z-direction to the positive z-direction in the half-plane with $r > 0$. These rays are sampled from an anisotropic hemispherical distribution in wave vector space, so that rays are more heavily weighted toward the r-direction rather than the $\pm z$ directions. In 3D this would correspond to a spherical distribution of rays such that each ray subtends approximately the same solid angle in wave vector space.

  Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based

on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is π/3 radians. Unlike the **Spherical** distribution, the rays in the **Conical** distribution each subtend the same angle in 2D, not the same solid angle in 3D.

For **Hemispherical** and **Conical**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudo-random numbers.

Illuminated Surface

Use the **Illuminated Surface** node to release rays under the assumption that the emitted rays are specularly reflected from an external radiation source. It is assumed that the entire selected surface has direct line of sight to the external radiation source; that is, shadowing due to other geometric entities is not taken into account.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

See Inlet for information on the following sections: **Initial Position**.

RAY DIRECTION VECTOR

Select the **Rays to release**: **Reflected** (the default) or **Refracted**.

- For **Reflected** select the **Reflection coefficient specification**: **Using absorption coefficient** (the default) or **Using refractive indices**.
- For **Refracted** enter a value for **Refractive index of exterior domains** n_{ext} (dimensionless). The default is 1.

Select an **Incident ray direction vector**: **User-defined direction** (the default), **User-defined point source**, or **Solar radiation** (3D components only).

Then based on the selection enter the applicable information:

- For **User-defined direction** enter the components of the **Incident ray direction vector** $\mathbf{L}_i$ (dimensionless) based on space dimension.
- For **User-defined point source** enter the **Point source location** $\mathbf{r}_{\mathrm{src}}$ (SI unit: m).
- For **Solar radiation** see Solar Radiation for information about the options.

ANGULAR PERTURBATIONS

If **User-defined direction** or **Solar radiation** is selected from the **Incident ray direction vector** list, you can include small perturbation factors designed to take the finite size of the sun (or another light source) into account. Select an option from the **Corrections for finite source diameter** list: **None** (the default), **Create light cones at release points**, or **Sample from conical distribution**.

- For **None** no perturbation to the initial ray direction is applied.
- For **Create light cones at release points** a cone-shaped distribution of rays is released at each point. Enter a value for the **Number of rays in wave vector space** (dimensionless) N_w (dimensionless). The default is 50. A cone-shaped distribution containing N_w rays is released at each point, which may cause a very large number of rays to be released.
- For **Sample from conical distribution** a single ray is released at each release point. This ray is given a random perturbation with uniform probability density within a cone-shaped distribution in wave vector space.

For **Create light cones at release points** and **Sample from conical distribution**, specify the **Maximum disc angle** Ψ_m (SI unit: rad). The default is 4.65 mrad, approximately the size of the solar disc when viewed from Earth.

For **Create light cones at release points** and **Sample from conical distribution**, select a **Limb darkening model**: **None** (the default), **Empirical power law**, **Linear**, or **User defined**. The limb darkening model is typically used to reduce the intensity of solar radiation that is released from the periphery of the solar disk, relative to radiation that is released near the center. For **Linear** enter a **Limb darkening coefficient** β (dimensionless). The default value is 0.8. For **User defined** enter a **Limb darkening coefficient** f_L (dimensionless). The default value is 1.

Select the **Include surface roughness** check box to include an additional perturbation term based on uncertainty in the orientation of the surface normal, which is common in imperfect reflecting surfaces. Enter a value or expression for the **Surface slope error** σ_ϕ (SI unit: rad). The default is 1 mrad. The surface normal is then perturbed by a random angle, which is sampled from a Rayleigh distribution.

INCIDENT RAY INTENSITY

This section is available when the ray intensity is solved for. Enter a value for the **Incident ray intensity** I_i (SI unit: W/m^2). The default is 1000 W/m^2.

TOTAL SOURCE POWER

This section is available:

- when the ray intensity is solved for, and
- when **Spherical**, **Hemispherical**, or **Conical** is selected as the **Ray direction vector**.
- It is also available when the ray power is solved for, and then any choice of **Ray direction vector** displays this section.

Enter a **Total source power** P_{src} (SI unit: W). The default is 1 W. In 2D, instead enter the **Total source power per unit thickness** P_{src} (SI unit: W/m). The default is 1 W/m.

Illuminated Surface Theory

Grating

Use the **Grating** node to treat a boundary as a diffraction grating that can release reflected and transmitted rays of multiple diffraction orders. A Diffraction Order subnode for reflected and transmitted rays of order $m = 0$ is added by default. Change the settings for this default subnode to release rays of a different diffraction order. You can also release rays of multiple diffraction orders from the same boundary by adding more **Diffraction Order** subnodes from the from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

The Accumulator (Boundary) subnode is also available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Releasing Secondary Rays

When two or more rays are released from a **Grating** boundary, one of the released rays uses the same degrees of freedom as the incident ray. The remaining degrees of freedom must be preallocated in memory as secondary rays. The total number of secondary rays that can be released in the model is controlled by the **Maximum number of secondary rays** field in the physics interface **Ray Release and Propagation** section.

For example, if both reflected and transmitted rays of diffraction orders $m = -1$, $m = 0$, and $m = 1$ are released from a boundary, then for every incident ray, a total of five secondary rays are released. If the diffraction order $m = -1$ is the first subnode to appear in the Model Builder, then the transmitted ray of order $m = -1$ uses the same degree of freedom as the incident ray, and the other rays are secondary rays. If the **Maximum number of secondary rays** is 500 and more than 100 rays interact with the grating, then no more secondary rays are emitted and a warning is generated by the solver, indicating that the **Maximum number of secondary rays** should be increased.

DEVICE PROPERTIES

Select an option from the **Rays to release** list: **Reflected and transmitted** (the default), **Reflected**, or **Transmitted**. Then enter the a **Grating constant** d (SI unit: m). The default is 600 nm.

If the ray power is solved for, the **Store total transmitted power** and **Store total reflected power** check boxes are shown. Select these check boxes to declare auxiliary dependent variables for the total power of all transmitted and reflected diffraction orders, respectively.

GRATING ORIENTATION

> This section is only available in 3D. In 2D, the lines of the grating are always assumed to point directly out of the plane, as shown in Figure 3-4.

Select an option from the **Grating orientation list**: **Specify direction of grating lines** (the default) or **Specify direction of periodicity**.

- If **Specify direction of grating lines** is selected, select an option from the **Direction of grating lines** list: **User defined** (the default) or **Parallel to reference edge**. For **User defined** enter the components of the direction of grating lines $\mathbf{T}_l$ (dimensionless) directly. The default is the positive x-axis. For **Parallel to reference edge**, the **Reference Edge Selection** section is shown. Select a single edge, which must be adjacent to at least one boundary in the selection for the **Grating** feature.
- If **Specify direction of periodicity** is selected, select an option from the **Direction of periodicity** list: **User defined** (the default) or **Parallel to reference edge**. For **User defined** enter the components of the direction of periodicity $\mathbf{T}_p$ (dimensionless) directly. The default is the positive x-axis. For **Parallel to reference edge**, the **Reference Edge Selection** section is shown. Select a single edge, which must be adjacent to at least one boundary in the selection for the **Grating** feature.

The vectors $\mathbf{T}_l$ and $\mathbf{T}_p$ are illustrated in Figure 3-5, along with typical paths for incident reflected, and transmitted rays.

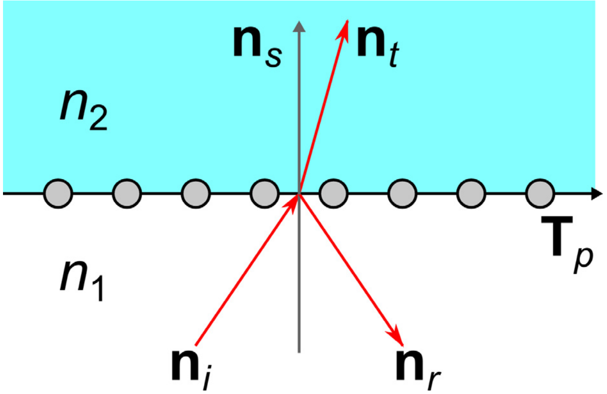


Figure 3-4: Diagram of diffraction grating orientation in 2D.

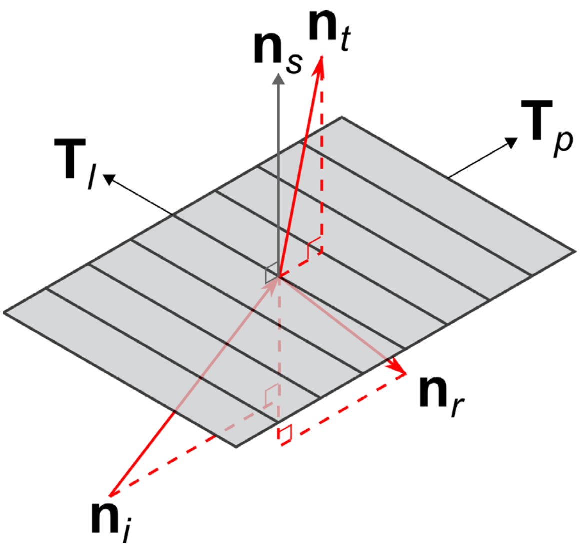


Figure 3-5: Diagram illustrating the options to specify grating orientation in 3D.

Grating Theory

Diffraction Order

Add the **Diffraction Order** subnode to the Grating feature. Each **Diffraction Order** subnode can either release transmitted rays, reflected rays, or both transmitted and reflected rays of the specified order, depending on the option selected from the **Rays to release** list in the parent **Grating** node.

DEVICE PROPERTIES

Enter a **Diffraction order** m (dimensionless). The default is 0 for the first **Diffraction Order** added. When more diffraction orders are added to the model, the new default is the highest diffraction order currently present, plus 1.

When the ray intensity or power is solved for, enter the **Reflectance** R and the **Transmittance** T (both dimensionless). If the reflectance or transmittance of a certain diffraction order is exactly zero, that diffraction order will not be released.

AUXILIARY DEPENDENT VARIABLES, REFRACTED RAY

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Material Discontinuity feature.

AUXILIARY DEPENDENT VARIABLES, REFLECTED RAY

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Material Discontinuity feature.

Linear Polarizer

Use the **Linear Polarizer** node to define a linear polarizer on a boundary selection. Add this node from the **Optical Devices** submenu when ray intensity or power is solved for in the model. The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

DEVICE PROPERTIES

Select a **Transmission axis specification**:

- **User defined** (the default). Then enter coordinates for the **Transmission axis** $\mathbf{T}$ (dimensionless).
- **Parallel to reference edge** (for 3D components). The **Reference Edge Selection** section is then shown. Add an edge to this selection to specify the transmission axis

orientation. Only a single edge may be selected. The edge must be adjacent to a boundary in the physics feature selection.

- **In-plane** or **Out-of-plane** (for 2D and 2D axisymmetric components)

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Wall feature.

Linear Polarizer Theory

Ideal Depolarizer

Use the **Ideal Depolarizer** node to reinitialize the Stokes parameters so that the outgoing ray is completely unpolarized. Add this node from the **Optical Devices** submenu when ray intensity or power is solved for in the model. The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

Ideal Depolarizer Theory

Linear Wave Retarder

Use the **Linear Wave Retarder** node to define a linear wave retarder on a boundary selection. Add this node from the **Optical Devices** submenu when ray intensity or power is solved for in the model. The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

DEVICE PROPERTIES

Select a **Fast axis specification**:

- **User defined** (the default). Then enter coordinates for the **Fast axis** $\mathbf{F}$ (dimensionless).
- **Parallel to reference edge** (for 3D components). The **Reference Edge Selection** section is then shown. Add an edge to this selection to specify the fast axis orientation. Only a single edge may be selected. The edge must be adjacent to a boundary in the physics feature selection.
- **In-plane** or **Out-of-plane** (for 2D and 2D axisymmetric components)

For any selection, enter a value for the **Retardance** δ (SI unit: rad).

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Wall feature.

Linear Wave Retarder Theory

Circular Wave Retarder

Use the **Circular Wave Retarder** node to define a circular wave retarder on a boundary selection. Add this node from the **Optical Devices** submenu when ray intensity or power is solved for in the model. The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

DEVICE PROPERTIES

Enter a value for the **Retardance** δ (SI unit: rad).

By default the **Sense of rotation depends on ray trajectory** check box is not selected. This means a positive value of the retardance always corresponds to clockwise rotation of the polarization ellipse. Select the check box then a positive value of the retardance corresponds to clockwise rotation of the polarization ellipse only when the ray crosses the boundary in the direction indicated by the normal vector in the Graphics window.

The **Reverse sense of rotation** check box is available when the **Sense of rotation depends on ray trajectory** check box is selected. Click to select this check box to reverse the direction of the normal vector shown in the Graphics window.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Wall feature.

Circular Wave Retarder Theory

Mueller Matrix

Use the **Mueller Matrix** node to define Mueller matrices for customized optical components. Add this node from the **Optical Devices** submenu when ray intensity or power is solved for in the model. The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

DEVICE PROPERTIES

Select a **Reference axis specification**:

- **User defined** (the default). Then enter coordinates for the **Reference axis** $\mathbf{R}$ (dimensionless).
- **Parallel to reference edge** (for 3D components). The **Reference Edge Selection** section is then shown. Add an edge to this selection to specify the reference direction orientation. Only a single edge may be selected. The edge must be adjacent to a boundary in the physics feature selection.
- **In-plane** or **Out-of-plane** (for 2D and 2D axisymmetric components).

For any choice, select **Isotropic** (the default), **Diagonal**, **Symmetric**, or **Anisotropic** from the list and then enter a value or expression for the **Mueller matrix** $\mathbf{M}$ in the matrix or field.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model. The settings are the same as for the Wall feature.

Theory of Mueller Matrices and Optical Components

Auxiliary Dependent Variable

Use the **Auxiliary Dependent Variable** node to solve additional first-order ordinary differential equations for each ray released.

AUXILIARY DEPENDENT VARIABLE

Enter a **Field variable name**. The default is `rp` and can be changed to anything provided it does not conflict with the name of the variables for the position or momentum degrees of freedom. The name should not conflict with other auxiliary dependent variables.

Enter a **Source** R. The unit of the source depends on the unit of the auxiliary dependent variable and the option selected from the **Integrate** list described below.

Select a way to **Integrate** the equation you have defined: **With respect to time** or **Along ray trajectory**. If you are interested in the residence time of a group of rays in a given system then you can set the **Source** to 1 and set **Integrate** to **With respect to time**. To compute the length of the ray trajectory you can set the **Source** to 1 and set **Integrate** to **Along ray trajectories**.

UNITS

These settings are the same as for Accumulator (Domain).

Release from Edge

Use the **Release from Edge** node to release rays from positions along a set of edges in a 3D geometry.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position**: **Mesh based** (the default), **Uniform distribution**, or **Density**. Mesh based and Density have the same settings as described for the Release node. If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected edges is divided into N segments of approximately equal length, and a ray is released from the middle of each segment.

Release from Point

Use the **Release from Point** node to release rays from a set of points in a geometry. By default one ray is released at every selected point, although it is possible to release multiple rays at each point if the initial direction or an auxiliary dependent variable is sampled from a distribution.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

Release from Point on Axis

The **Release from Point on Axis** feature is similar to the Release from Point feature, except that it can only release rays from points along the axis of symmetry in a 2D axisymmetric geometry.

See Inlet for information on the following sections: **Ray Direction Vector**.

For all other sections, see Release from Point.

Release from Grid

Use the **Release from Grid** node to release rays from a user-defined grid of points.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, **Initial Polarization**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL COORDINATES

Select an option from the **Grid type** list: **All combinations** (the default), **Specified combinations**, **Cylindrical** (3D only), or **Hexapolar** (3D only).

Linear, Rectangular, and Arbitrary Grids

For **All combinations** and **Specified combinations** enter **Initial coordinates** based on space dimension ($q_{x,0}$, $q_{y,0}$, and $q_{z,0}$ for 3D components) for the ray positions or click the **Range** button (⋯) to select and define a range of specific coordinates.

If **Specified combinations** is selected, the number of initial coordinates entered for each space dimension must be equal, and the total number of rays released is equal to the length of one of the lists of initial coordinates. If **All combinations** is selected, the total number of rays released is equal to the product of the lengths of each list of initial coordinates.

For example, suppose a 2D component includes a **Release from Grid** node with the following initial coordinates:

- $q_{x,0}$ = range(0,1,3)
- $q_{y,0}$ = range(2,2,8)

If **All combinations** is selected, a total of 16 rays will be released, including every possible combination of the initial x- and y-coordinates. If **Specified combinations** is selected, 4 rays will be released with initial positions (0,2), (1,4), (2,6), and (3,8).

Cylindrical Grids

For **Cylindrical**, enter coordinates for the **Center location** $\mathbf{q}_c$ (SI unit: m). By default, the distribution is centered at the origin. Then enter the components of the **Cylinder axis direction** $\mathbf{r}_c$ (dimensionless). The rays will be released at specified radial distances and angles in the plane containing the point $\mathbf{q}_c$ and orthogonal to the direction $\mathbf{r}_c$.

Select an option from the **Radial distribution** list: **Uniform radius intervals** (the default), **Uniform number density**, or **User defined**.

For **Uniform radius intervals** or **Uniform number density** enter a value or expression for the **Radius** R_c (SI unit: m). The default is 1 m. Then enter a positive integer for the **Number of radial positions** N_c (dimensionless). The default is 5.

For **User defined** enter a list of **Radial coordinates** q_r (SI unit: m) directly. An arbitrary number of radial coordinates can be entered in the list. The default is 1 m.

The effect of the **Radial distribution** setting on the resulting ray placement is illustrated in Figure 3-6. For **Uniform radius intervals**, the distances between the concentric rings of rays are all equal, but the number density of rays is greater at the center of the distribution than at the outer edge. The option **Uniform number density** corrects this imbalance by defining nonuniform increments in the radial position between the concentric rings.

Enter a positive integer for the **Number of angles** N_ϕ (dimensionless). The default is 10.

The total number of rays released (before accounting for distributions of ray direction or auxiliary dependent variables at each release point) is $N_c \times N_\phi + 1$ because a single ray is also released at the center of the distribution.

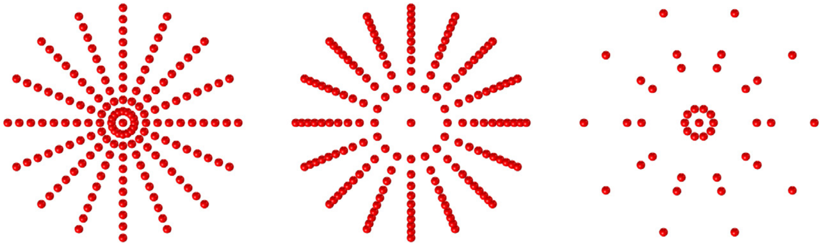


Figure 3-6: Comparison of the cylindrical release in which rays are positioned at uniform radius intervals (left), with uniform number density (center), or at user-defined radii (right).

Hexapolar Grids

For **Hexapolar**, enter coordinates for the **Center location** $\mathbf{q}_c$ (SI unit: m). By default, the distribution is centered at the origin. Then enter the components of the **Cylinder axis direction** $\mathbf{r}_c$ (dimensionless). The rays will be released at specified radial distances and angles in the plane containing the point $\mathbf{q}_c$ and orthogonal to the direction $\mathbf{r}_c$.

Enter a value or expression for the **Radius** R_c (SI unit: m). The default is 1 m. Then enter a positive integer for the **Number of radial positions** N_c (dimensionless). The default is 5.

Unlike the **Cylindrical** grid types described in the previous section, each concentric ring of the **Hexapolar** grid releases a different number of rays, as shown in Figure 3-7. One ray is always released at the center. The first ring of rays surrounding the center has 6 rays arranged in a regular hexagon. Each ring of rays beyond the first has 6 more rays than the ring preceding it, with the grid points arranged in a regular polygon. The radius increments between consecutive rings are uniform.

The total number of grid points generated is

$$N_{\mathrm{grid}} = 1 + 3N_c(N_c + 1)$$

Thus the relationship between the number of radius intervals and the total number of grid points is quadratic.

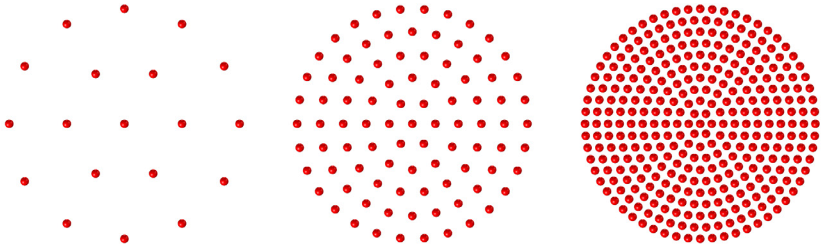


Figure 3-7: Comparison of hexapolar grids with 2 rings (left), 5 rings (middle), and 10 rings (right).

Release from Grid on Axis

The **Release from Grid on Axis** feature is similar to the Release from Grid feature, except that it can only release rays from a grid of points along the axis of symmetry in a 2D axisymmetric geometry; that is, $q_{r,0} = 0$ for all released rays.

See Inlet for information on the following sections: **Ray Direction Vector**.

For all other sections, see Release from Grid.

Release from Data File

Use a **Release from Data File** node to specify the initial ray position, direction, and values of auxiliary dependent variables using data from a text file.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Polarization**.

Formatting Guidelines for Files Containing Ray Data

The imported data file should be a text file (`*.txt`) arranged in a spreadsheet format; that is, each row corresponds to a distinct ray and should have the same number of columns as all other rows. Columns can be separated by spaces, tabs, or a combination of the two. Begin a line with the percent (“`%`”) character to include comments or empty lines in the data file.

For example, a data file containing the following text would release rays at the positions (0.1, 0.2, 0.6) and (0.2, 0.4, 0.8) in a three-dimensional geometry:

```
% Ray release positions
% qx0 qy0 qz0
0.1 0.2 0.6
0.2 0.4 0.8
```

INITIAL POSITION

Browse your computer to select a text file, then click **Import** to import the data. To remove the imported data, click **Discard**. Enter the **Index of first column containing position data** i to indicate which column represents the first coordinate of the ray position vectors. The default value, 0, indicates the first column.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **From file**, **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **From file**, enter the **Index of first column containing velocity data** i. The default is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.
- For all other settings, the settings are the same as for the Release node.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the active **Auxiliary Dependent Variable** nodes in the model, choose an option from the **Distribution function** list: **From file**, **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of Values**.

- When **From file** is selected, enter the **Index of column containing data**. The default value is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.

For all other options, the settings are the same as for the Release node.

Ray Continuity

Use the **Ray Continuity** node to specify that rays should cross a pair boundary as if it were invisible. Pair boundaries appear when the geometry sequence ends in **Form Assembly** instead of **Form Union**. Such boundaries require special handling because the mesh elements on either side of the pair boundary aren’t required to match up exactly.

PAIR SELECTION

Select one or more identity pairs to allow rays to cross between the source and destination boundaries of these pairs. Such identity pairs are typically created automatically on interior boundaries when the geometry sequence ends in a **Form Assembly** node instead of a **Form Union** node.

The **Ray Continuity** node does not cause the ray position components to change discontinuously; the source and destination boundaries for the identity pairs must be overlapping.

If the geometry sequence ends in a **Form Union** node, the **Ray Continuity** node usually isn't necessary because rays can freely cross interior boundaries where no boundary condition has been applied.

Solar Radiation

The **Solar Radiation** node uses the solar position to specify the initial direction vector of rays. The node is selected from the **Physics** toolbar, **Global** menu.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Vacuum Wavelength**, **Initial Phase**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL COORDINATES

These settings are the same as for the Release from Grid node.

North, West, and the up directions correspond to the *x*, *y*, and *z* directions, respectively.

SOURCE

Select an option from the **Location defined by** list: **Coordinates** (the default) or **City**.

For **City** select a predefined city and country combination from the list. Click to select the **Include daylight saving time (Time zone + 1)** check box to add one hour to the default setting for the city selected. For example, if **New York City, USA** is selected and the default standard time zone is UTC -5 hours. If the check box is selected, the daylight savings time is used instead (UTC -4 hours). This feature doesn't automatically deduce whether daylight savings should be applied based on the selected city and date; you must specify it explicitly.

If **Coordinates** is selected, or your city is not listed, in the **Location** table define the:

- **Latitude**, a decimal value, positive in the northern hemisphere (the default is Las Vegas, USA latitude, 36.1). Enter a value without a unit to avoid double conversion. This is because the latitude value is expected to represent degrees but the model's unit for angles may be different (for example, the SI unit for the angle is radians).
- **Longitude**, a decimal value, positive at the East of the Prime Meridian (the default is Las Vegas, USA longitude, -115.2). Enter a value without a unit to avoid double conversion. This is because the latitude value is expected to represent degrees but the model's unit for angles may be different (for example, the SI unit for the angle is radians).
- **Time zone**, the number of hours to add to UTC to get local time (the default is Las Vegas, USA time zone, -8). For example in **New York City, USA** the time zone is UTC -5 hours (standard time zone) or UTC -4 hours (with daylight savings time).

For either selection, in the **Date** table enter the:

- **Day**, the default is 01. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent days but the model's unit for time may be different (for example, the SI unit for time is second).
- **Month**, the default is 10 (October). Enter a value without a unit to avoid double conversion. This is because the value is expected to represent months but the model's unit for time may be different (for example, the SI unit for time is second).
- **Year**, the default is 2014. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent years but the model's unit for time may be different (for example, the SI unit for time is second). The solar position is accurate for a date between 2000 and 2199.

For either selection, in the **Local time** table enter the:

- **Hour**, the default is 12. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent hours but the model's unit for time may be different (for example, the SI unit for time is second).

- **Minute**, the default is 0. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent minutes but the model's unit for time may be different (for example, the SI unit for time is second).
- **Second**, the default is 0.

The sun position is updated if the location, date, or local time changes during a simulation. Because the time scale of ray propagation is typically many orders of magnitude smaller than the time scale for significant changes in solar position, modeling ray propagation at different times of day typically requires a **Parametric Sweep** over the larger time scale. Note that no validity range is prescribed on the time inputs. It is possible to enter values that exceed the expected boundary. For example, entering 5h 2min 81s is equivalent to 5h 3min 21s. This makes it easier, for example, to run a Parametric Sweep over the time of day using only a single parameter.

For either selection, in the **Solar irradiance** field I_s (SI unit: $\mathrm{W/m^2}$) define the incident radiative intensity coming from the sun.

ANGULAR PERTURBATIONS

Select an option from the **Corrections for finite source diameter** list: **None** (the default), **Create light cones at release points**, or **Sample from conical distribution**.

- For **None** no perturbation to the initial ray direction is applied.
- For **Create light cones at release points** a cone-shaped distribution of rays is released at each point. Enter a value for the **Number of rays in wave vector space** (dimensionless) N_w (dimensionless). The default is 50. A cone-shaped distribution containing N_w rays is released at each point, which may cause a very large number of rays to be released.
- For **Sample from conical distribution** a single ray is released at each release point. This ray is given a random perturbation with uniform probability density within a cone-shaped distribution in wave vector space.

For **Create light cones at release points** and **Sample from conical distribution**, specify the **Maximum disc angle** Ψ_m (SI unit: rad). The default is 4.65 mrad.

For **Create light cones at release points** and **Sample from conical distribution**, select a **Limb darkening model**: **None** (the default), **Empirical power law**, **Linear**, or **User defined**. The limb darkening model is typically used to reduce the intensity of solar radiation that is released from the periphery of the solar disk, relative to radiation that is released near the center. For **Linear** enter a **Limb darkening coefficient** β (dimensionless). The default value is 0.8. For **User defined** enter a **Limb darkening coefficient** f_L (dimensionless). The default value is 1.

The options for accounting for finite source diameter and solar limb darkening are discussed in the Illuminated Surface Theory section.

Ray Termination

Use the **Ray Termination** feature to remove rays from the model as they exit the geometry, or to annihilate rays of sufficiently low intensity or power. Only one **Ray Termination** node can be added to the model, but it is possible to combine position-based and intensity-based criteria for removing rays if the ray intensity or power is computed by the model.

TERMINATION CRITERIA

Select an option from the **Spatial extents of ray propagation** list: **Bounding box, from geometry** (the default), **Bounding box, user defined**, or **None**. This list controls whether rays are terminated based on their positions.

- For **Bounding box, from geometry**, a ray is immediately terminated if it reaches the surface of the bounding box of the geometry, defined as the smallest rectangle or rectangular prism with edges parallel to the axes of the global coordinate system that encloses all of the geometric entities. This bounding box is extended by 5% in each direction so that rays can still interact with boundaries that coincide with the perimeter of the bounding box. Therefore rays can propagate a short distance away from the geometry before disappearing.
- For **Bounding box, user defined**, a ray is immediately terminated if it reaches the surface of a box of user-specified dimensions. To define the box, enter maximum and minimum values of each spatial coordinate.
- For **None**, the rays that propagate outward from the geometry will continue to propagate until the study is complete. If the maximum path length for the **Ray Tracing** study step is too large, this may result in trajectories that are extremely long relative to the characteristic geometry size.

If rays are released from outside the bounding box, they won’t immediately disappear; instead, they will only disappear if they enter the bounding box and then exit it again.

Intensity- and Power-Based Termination Criteria

If the ray intensity or power is being computed in the model, you can then terminate rays based on either or both of these quantities.

Select an option from the **Additional termination criteria** list. **None** (the default) is always available. If ray intensity is solved for, then **Intensity** is available. If ray power is solved for, then **Power** is available. If both ray intensity and power are solved for, the **Intensity and Power** option is also shown.

- For **Intensity** or **Intensity and power**, enter the **Threshold intensity** I_{th} (SI unit: $\mathrm{W/m^2}$). The default is `1e-3[W/m^2]`. Rays will be removed from the model if their intensity decreases below the specified value.
- For **Power** or **Intensity and power**, enter the **Threshold power** Q_{th} (SI unit: W). The default is `1e-3[mW]`.

The **Ray Termination** feature can terminate rays at the exact time at which their intensity or power reaches the specified threshold (to within numerical precision), if there exists a closed-form analytic expression for this time. Such a closed-form analytic solution exists if:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a non-absorbing medium.

Otherwise, the rays are stopped at the first discrete time step taken by the solver for which the intensity or power is less than the threshold value.

Ray Termination Theory

Ray Detector

Use the **Ray Detector** feature to compute information about rays that are located in a set of domains or on a set of selected boundaries. The detector may detect all rays or only the rays released by a specified release feature. Computed variables are the number of rays transmitted, the transmission probability, and a logical expression which can be used to filter the rendered rays during results processing.

The **Ray Detector** feature is also capable of computing the total transmitted ray power. If the ray power is solved for, then a global variable for the total power transmitted to the detector is available.

RAY DETECTOR

Select an option from the **Release feature** list: **All** (the default), or any of the features which are capable of releasing rays. When **All** is selected, the variables computed are summed over all release features in the model.

Theory for the Geometrical Optics Interface

The Geometrical Optics Interface theory is described in this section:

Introduction to Geometrical Optics

Assume that the electric field **E** of an electromagnetic wave can be represented in the form

$$\mathbf{E} = \mathbf{a}e^{i\Psi}$$

where $\mathbf{a}$ is the slowly varying amplitude and the phase Ψ is a function of the position vector $\mathbf{q}$ and time t. If the field is observed only at locations that are very far from any source, the phase can be expressed as

$$\Psi = \mathbf{k} \cdot \mathbf{q} - \omega t + \alpha$$

where $\mathbf{k}$ is the wave vector, ω is the angular frequency, and α is an arbitrary phase shift. In an isotropic medium, the wave vector and the angular frequency are further related by the expression

$$\omega = \frac{c|\mathbf{k}|}{n(\mathbf{q})}$$

where $n(\mathbf{q})$ is the refractive index of the medium. It follows that the wave vector and angular frequency can be expressed in terms of the phase:

$$\mathbf{k} = \frac{\partial\Psi}{\partial\mathbf{q}} \qquad \omega = -\frac{\partial\Psi}{\partial t}$$

Following Landau and Lifshitz in Ref. 1, the wave vector and frequency are analogous to the generalized momentum $\mathbf{p}$ and Hamiltonian H of a solid particle,

$$\mathbf{p} = \frac{\partial S}{\partial\mathbf{q}} \qquad H = -\frac{\partial S}{\partial t}$$

where S is the integral of the Lagrangian along the particle's trajectory. From this analogy, it follows that the ray trajectory can be computed by solving six coupled first-order ordinary differential equations for the components of $\mathbf{k}$ and $\mathbf{q}$:

$$\frac{d\mathbf{k}}{dt} = -\frac{\partial\omega}{\partial\mathbf{q}} \qquad \frac{d\mathbf{q}}{dt} = \frac{\partial\omega}{\partial\mathbf{k}}$$

In regions of constant refractive index, the simplified equations of motion are:

$$\frac{d\mathbf{k}}{dt} = \mathbf{0} \qquad \frac{d\mathbf{q}}{dt} = \frac{c\mathbf{k}}{n|\mathbf{k}|}$$

These first-order equations for the ray trajectory are only valid in regions that are many wavelengths away from any point source; that is, $\Psi » 1$.

Optical Dispersion Models

The **Refractive index, real part**, n (dimensionless) may be expressed as a function of wavelength using an optical dispersion model. Each model requires a set of coefficients to be specified. These can be determined, for example, by making a least squares fit to a set of refractive index measurements at discrete wavelengths.

At least one optical dispersion model (**Temperature-dependent Sellmeier**) allows the refractive index to be expressed as a function of both wavelength and temperature. All other optical dispersion models may be used together with a thermo-optic dispersion model to calculate temperature-dependent indices. This is discussed in Thermo-Optic Dispersion Models.

RELATIVE VS. ABSOLUTE REFRACTIVE INDICES

It is important to appreciate that the refractive indices given by manufacturers are often relative to air at a certain temperature and pressure. Therefore, each of the optical dispersion models (except **Temperature-dependent Sellmeier)** require a reference temperature T_{ref} (SI unit: K) and pressure P_{ref} (SI unit: Pa) to be specified. This is interpreted to be the temperature and pressure of the air in which the refractive indices used to generate the model were measured. The reference pressure is absolute. If the reference pressure $P_{\mathrm{ref}} \neq 0$, then it is assumed that the dispersion model will give relative refractive indices n_{rel} (dimensionless). The conversion of refractive index from relative (n_{rel}) to absolute (n) is made using a model for the refractive index of air (n_{air}). That is,

$$n = n_{\mathrm{rel}} \times n_{\mathrm{air}} .$$

The model, described in Ref. 2, used to calculated the refractive index of air is

$$n_{\mathrm{air}}(\lambda, T, P) = \frac{n_{\mathrm{air,\,STP}}(\lambda) - 1}{1 + 3.4785\times10^{-3}(T - T_{\mathrm{std}})}\frac{P}{P_{\mathrm{std}}} ,$$

where $T = T_{\mathrm{ref}}$ and $P = P_{\mathrm{ref}}$ and $n_{\mathrm{air,STP}}$ is the refractive index of air at standard temperature and pressure. By definition, the standard temperature is $T_{\mathrm{std}} = 15$ °C and the standard pressure is $P_{\mathrm{std}} = 101{,}325$ Pa and then (with a CO_2 concentration of 0.03%) $n_{\mathrm{air,STP}}$ is given by

$$n_{\mathrm{air,\,STP}}(\lambda) = 1 + 10^{-8} \times \left(6432.8 + \frac{2{,}949{,}810\lambda^2}{146\lambda^2 - 1} + \frac{25{,}540\lambda^2}{41\lambda^2 - 1}\right) .$$

OPTICAL DISPERSION MODELS

The built-in optical dispersion models are listed in Table 3-3.

TABLE 3-3: LIST OF OPTICAL DISPERSION MODELS

NAME	EXPRESSION
Cauchy	$n_{\mathrm{rel}} = A + \frac{B}{\lambda^2} + \frac{C}{\lambda^4}$
Conrady	$n_{\mathrm{rel}} = A + \frac{B}{\lambda} + \frac{C}{\lambda^{3.5}}$
Herzberger	$n_{\mathrm{rel}} = A_0 + \frac{A_1}{\lambda^2 - 0.028} + \frac{A_2}{(\lambda^2 - 0.028)^2} + A_3\lambda^2 + A_4\lambda^4$
Schott	$n_{\mathrm{rel}}^2 = A_0 + A_1\lambda^2 + \frac{A_2}{\lambda^2} + \frac{A_3}{\lambda^4} + \frac{A_4}{\lambda^6} + \frac{A_5}{\lambda^8}$
Schott extended	$n_{\mathrm{rel}}^2 = A_0 + A_1\lambda^2 + A_2\lambda^4 + \frac{A_3}{\lambda^2} + \frac{A_4}{\lambda^4} + \frac{A_5}{\lambda^6} + \frac{A_6}{\lambda^8} + \frac{A_7}{\lambda^{10}} + \frac{A_8}{\lambda^{12}}$
Sellmeier	$n_{\mathrm{rel}}^2 = 1 + \frac{A_1\lambda^2}{\lambda^2 - B_1} + \frac{A_2\lambda^2}{\lambda^2 - B_2} + \frac{A_3\lambda^2}{\lambda^2 - B_3}$
Sellmeier modified, type 1	$\frac{n_{\mathrm{rel}}^2 - 1}{n_{\mathrm{rel}}^2 + 2} = \frac{A_1\lambda^2}{\lambda^2 - B_1} + \frac{A_2\lambda^2}{\lambda^2 - B_2} + \frac{A_3\lambda^2}{\lambda^2 - B_3}$
Sellmeier modified, type 2	$n_{\mathrm{rel}}^2 = A + \frac{B_1\lambda^2}{\lambda^2 - C_1} + \frac{B_2\lambda^2}{\lambda^2 - C_2}$
Temperature - dependent Sellmeier	$n^2 = 1 + \sum_{i=1}^{3} \frac{S_i(T)\lambda^2}{\lambda^2 - \lambda_i^2(T)},\ S_i(T) = \sum_{j=0}^{4} S_{ij}T^j,\ \lambda_i(T) = \sum_{j=0}^{4} \lambda_{ij}T^j$

All optical dispersion models assume that coefficients have been determined with expressions where the wavelengths have units of μm. For example, in the Schott model, the coefficients have units 1, μm, μm^2, μm^3, and so on. The input wavelength can still have any unit; the normalization to microns is done automatically.

The **Temperature-dependent Sellmeier** model (Ref. 3, Ref. 4) assumes coefficients based on absolute temperature specified in Kelvin (K).

Thermo-Optic Dispersion Models

The **Refractive index, real part**, n (dimensionless) may be expressed as a function of both wavelength and temperature by using an expression of the form

$$n(\lambda,T) = n(\lambda) + \Delta n(\lambda,T) .$$

The wavelength-dependent (non-temperature-dependent) component of this equation, $n(\lambda)$, might be any one of the optical dispersion models discussed above, or it could be specified by other means. The change in refractive index as a function of temperature, $\Delta n(\lambda,\mathrm{T})$ can be computed using a thermo-optic dispersion model.

SCHOTT THERMO-OPTIC DISPERSION MODEL

An expression for the change in refractive index as a function of temperature, $\Delta n(\lambda,T)$ is given in Ref. 5. It may be derived by integrating the dispersion formula for the thermo-optic coefficient ($\mathrm{d}n/\mathrm{d}T$). That is,

$$\frac{\mathrm{d}n(\lambda,T)}{\mathrm{d}T} = \frac{n^2(\lambda,T_0)-1}{2n(\lambda,T_0)}\left(D_0 + 2D_1\Delta T + 3D_2\Delta T^2 + \frac{E_0 + 2E_1\Delta T}{\lambda^2 - \lambda_{\mathrm{TK}}^2}\right),$$

which gives

$$\Delta n(\lambda,T) = \frac{n(\lambda,T_0)-1}{2n(\lambda,T_0)}\left(D_0\Delta T + D_1\Delta T^2 + D_2\Delta T^3 + \frac{E_0\Delta T + E_1\Delta T^2}{\lambda^2 - \lambda_{\mathrm{TK}}^2}\right).$$

In the above formula T_0 is the reference temperature against which the temperature difference $\Delta T = T - T_0$ is computed and D_0, D_1, D_2, E_0, E_1, and λ_{TK}, are glass specific coefficients.

Initial Conditions: Direction

There are several options available for specifying the initial direction of rays. These options work by assigning values to the degrees of freedom corresponding to the wave vector $\mathbf{k}$ (SI unit: rad/m) of each ray.

EXPRESSION

The default is to enter an expression for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless). This vector need not have a magnitude of unity because it is always normalized automatically. The initial wave vector is

$$\mathbf{k} = \frac{\omega n \mathbf{L}_0}{c|\mathbf{L}_0|}$$

where ω (SI unit: rad/s) is the angular frequency, $c = 299{,}792{,}458$ m/s is the speed of light in a vacuum, and n (dimensionless) is the refractive index.

SPHERICAL

When **Spherical** is selected the initial wave vectors are sampled from a distribution in wave vector space at each release point. The number of rays released from each point is usually equal to the specified value N_w (dimensionless), although it may be larger if the initial values of any auxiliary dependent variables are also sampled from a distribution.

In 2D the initial wave vector components are

$$k_x = \frac{\omega n}{c}\cos\theta$$

$$k_y = \frac{\omega n}{c}\sin\theta$$

where θ goes from 0 to 2π in N_w steps. In 3D the initial wave vector components are sampled according to the expressions

$$k_x = \frac{\omega n}{c}\cos\theta\sin\varphi$$

$$k_y = \frac{\omega n}{c}\sin\theta\sin\varphi$$

$$k_z = \frac{\omega n}{c}\cos\varphi$$

The azimuthal angle θ is uniformly distributed from 0 to 2π. The polar angle φ is sampled from the interval [0, π] with probability density proportional to sin φ. The polar angle is arbitrarily chosen as the angle that the initial wave vector makes with the positive z-axis, but any direction could be chosen because the sphere is isotropic. Therefore each ray subtends approximately the same solid angle in wave vector space.

HEMISPHERICAL

The **Hemispherical** option is the same as the **Spherical** option, except that in 2D θ goes from 0 to π and in 3D φ goes from 0 to π/2. The angle (θ in 2D or φ in 3D) is measured from the direction given by the **Hemisphere axis** setting.

CONICAL

The **Conical** option is the same as the **Spherical** option, except that in 2D θ goes from 0 to α and in 3D φ goes from 0 to α. The angle (θ in 2D or φ in 3D) is measured from the direction given by the **Cone axis** setting.

LAMBERTIAN

The **Lambertian** option releases rays within a hemisphere in 3D, but the probability distribution function is different from that of the **Hemispherical** option. Recall that for an isotropic hemispherical distribution the polar angle φ has a probability density proportional to $\sin\varphi$; for the Lambertian distribution the probability density is instead proportional to $\sin\varphi\cos\varphi$. Because of this extra cosine term, distributions following this probability density are said to follow Lambert's cosine law.

Material Discontinuity Theory

At a Material Discontinuity between two different media, the wave vector is reinitialized using Snell's law. First, the angle of incidence θ_i is computed:

$$\theta_i = \operatorname{acos}\left(\frac{\mathbf{n}_i\cdot\mathbf{n}_s}{|\mathbf{n}_i||\mathbf{n}_s|}\right)$$

where $\mathbf{n}_i$ is a unit vector in the direction of the incident ray and $\mathbf{n}_s$ is a unit vector normal to the material discontinuity.

At a boundary between two isotropic, non-absorbing media, the refracted ray propagates in the direction $\mathbf{n}_t$ given by the following relations:

$$\mathbf{n}_t = \eta\mathbf{n}_i + \gamma\mathbf{n}_s$$

$$\gamma = -\eta\cos\theta_i + \cos\theta_t$$

$$\eta = \frac{n_1}{n_2}$$

$$\theta_t = \operatorname{asin}(\eta\sin\theta_i)$$

where the ray propagates from the medium with refractive index n_1 into the medium with refractive index n_2. Because the two media are non-absorbing, the quantities n_1, n_2, θ_i, and θ_t are real-valued. If the media are weakly absorbing, the real parts of n_1 and n_2, are used. If the media are strongly absorbing, several corrections to Snell's law and the Fresnel equations must be made, as discussed in Refraction in Strongly Absorbing Media.

If the maximum number of secondary rays has not yet been released, a reflected ray is also released at the material discontinuity with the initial direction

$$\mathbf{n}_r = \mathbf{n}_i - 2\mathbf{n}_s \cos\theta_i$$

If the ray intensity is computed, the release of secondary rays may be suppressed when the intensity of the reflected ray is less than the user-defined **Threshold intensity** I_{th} (SI unit: W/m^2). Similarly, when ray power is computed, a **Threshold power** Q_{th} (SI unit: W) can be specified.

Intensity, Wavefront Curvature, and Polarization

The following describes the algorithm used to compute the ray intensity and polarization when **Intensity computation** is set to **Compute intensity** or **Compute intensity and power** in the physics interface **Intensity Computation** section. In this algorithm, each ray is treated as a propagating wavefront subtending a small solid angle. The algorithm is only valid when the rays propagate in homogeneous media. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The initial ray intensity I_0 and normalized Stokes parameters s_{n1}, s_{n2}, and s_{n3}. By allocating four degrees of freedom in this way it is possible to characterize any intensity magnitude and polarization state.
- The integral A of the attenuation coefficient along the ray path.
- Two initial principal radii of curvature, $r_{1,i}$ and $r_{2,i}$.
- Two principal radii of curvature, r_1 and r_2, which represent the maximum and minimum radii of curvature of the intersection of the wavefront with an arbitrary plane.
- Three components of a unit vector $\mathbf{e}_1$ in the direction corresponding to the first principal radius of curvature. This information is used to reinitialize the principal radii of curvature at curved boundaries.

In 2D:

- The initial ray intensity I_0 and normalized Stokes parameters s_{n1}, s_{n2}, and s_{n3}.
- The integral A of the attenuation coefficient along the ray path.

- One initial principal radius of curvature, $r_{1,i}$.
- One radius of curvature r_1 of the wavefront. All wavefronts are assumed to be converging or diverging cylindrical waves, so it is not necessary to define a second radius of curvature.

A different algorithm is used to compute the intensity and polarization of each ray if the **Intensity computation** is instead set to **Compute intensity in graded media** or **Compute intensity and power in graded media**. This algorithm accounts for the effect of graded media on the ray intensity, but it is also slower and less accurate for homogeneous media. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The intensity help variable Γ and the normalized Stokes parameters s_{n01}, s_{n02}, and s_{n03}, which characterize the intensity and polarization of the ray. They are reinitialized at material discontinuities and walls.
- The integral A of the attenuation coefficient along the ray path.
- Two principal curvature calculation help variables α_1 and α_2 and the rotation angle ϕ which indicates the orientation of the principal curvature directions. An additional help variable is used internally to detect poles in the local coordinate system definition and to redefine the local coordinate system accordingly.

In 2D:

- The intensity help variable Γ and the normalized Stokes parameters s_{n01}, s_{n02}, and s_{n03}, which characterize the intensity and polarization of the ray. They are reinitialized at material discontinuities and walls.
- The integral A of the attenuation coefficient along the ray path.
- One principal curvature calculation help variable α_1.

For the options **Compute Power**, **Compute intensity and power**, and **Compute intensity and power in graded media** an additional auxiliary dependent variable is defined to indicate the total power transmitted by each ray. Like intensity, power is affected by absorbing media and by reflection or refraction at boundaries. Unlike intensity, power doesn't change due to the focusing or divergence of a ray bundle.

THE STOKES PARAMETERS

The four Stokes parameters completely characterize the intensity and polarization of a fully polarized, partially polarized, or unpolarized ray. The Stokes parameters can be interpreted as indicators of the ray intensity that would be measured when sending a ray through various arrangements of polarizers and wave retarders.

The Stokes parameters of a ray are defined as in Ref. 6 as follows. Consider a ray propagating along the z-axis of a Cartesian coordinate system, with electric field components polarized within the xy-plane. In 3D models the x-axis is parallel to the first principal curvature direction $\mathbf{e}_1$. In 2D models the x-axis is always oriented in the out-of-plane direction.

Let $I(\theta,\varepsilon)$ be the intensity of radiation polarized in the xy-plane at an angle θ counterclockwise from the x-axis, when a phase angle of ε is introduced between the x- and y-components. For example, $I(0,0)$ is the intensity that would be measured after sending a ray through a linear polarizer with a transmission axis parallel to the x-axis, and $I(\pi/4,\pi/2)$ is the intensity that would be measured after sending a ray through a system of polarizers and wave retarders that only transmits radiation with right-handed circular polarization. The four Stokes parameters are then defined as follows:

- $s_0 = I(0, 0) + I(\pi/2, 0)$ is the sum of the intensity of linearly polarized light in the x- and y-directions. It represents the total intensity of the ray.
- $s_1 = I(0, 0) - I(\pi/2, 0)$ is the difference in intensity between light that is linearly polarized in the x-direction and light that is linearly polarized in the y-direction.
- $s_2 = I(\pi/4, 0) - I(3\pi/4, 0)$ is the difference in intensity between linearly polarized light in the direction of the line $y = x$ and linearly polarized light in the direction of the line $y = -x$.
- $s_3 = I(\pi/4, \pi/2) - I(3\pi/4, \pi/2)$ is the difference in intensity between light with right-handed circular polarization and light with left-handed circular polarization.

For example, a ray with linear polarization parallel to the x-axis has Stokes parameters $s_0 = s_1 = I$ and $s_2 = s_3 = 0$. A fully polarized ray is characterized by the relation

$$s_0^2 = s_1^2 + s_2^2 + s_3^2$$

A partially polarized or unpolarized ray is characterized by the relation

$$s_0^2 \geq s_1^2 + s_2^2 + s_3^2$$

The degree of polarization P is defined as

$$P = \frac{\sqrt{s_1^2 + s_2^2 + s_3^2}}{s_0}$$

A degree of polarization $P = 1$ corresponds to fully polarized radiation, whereas a degree of polarization $P = 0$ corresponds to unpolarized radiation.

In the COMSOL implementation of ray intensity calculation, the degrees of freedom are not the Stokes parameters themselves, but rather normalized Stokes parameters s_{n1}, s_{n2}, and s_{n3}. Then the actual Stokes parameters are recovered through the relations $s_1 = Is_{n1}$, $s_2 = Is_{n2}$, and $s_3 = Is_{n3}$. The reason for this decoupling is that the normalized Stokes parameters remain bounded, enabling accurate reinitialization of ray power, even if a ray is reflected or refracted very close to a caustic, where the intensity (in the geometrical optics limit) can become arbitrarily large.

PRINCIPAL RADII OF CURVATURE

In 3D models, each ray is treated as a wavefront for which two principal radii of curvature, r_1 and r_2, are defined. In addition, the values of the two principal radii of curvature are stored as $r_{1,i}$ and $r_{2,i}$ whenever the ray reaches a boundary.

Within domains, the equations

$$\frac{dr_1}{ds} = -1 \qquad \frac{dr_2}{ds} = -1$$

are solved. Negative radii of curvature indicate that the wavefront is expanding as the ray propagates, while positive radii of curvature indicate that the wavefront is converging. A continuous locus of points at which either of the principal radii of curvature equals zero is called a caustic surface. The unit vector $\mathbf{e}_1$ is defined so that r_1 is the radius of curvature of the intersection of the wavefront with the plane tangent to $\mathbf{e}_1$ and the wave vector $\mathbf{k}$. Similarly, if a vector $\mathbf{e}_2$ is defined so that

$$\mathbf{e}_2 = \frac{\mathbf{k} \times \mathbf{e}_1}{|\mathbf{k} \times \mathbf{e}_1|}$$

Then r_2 is the radius of curvature of the intersection of the wavefront with a plane tangent to $\mathbf{e}_2$ and $\mathbf{k}$. The components of $\mathbf{e}_1$ are stored as auxiliary dependent variables for each ray. The components of $\mathbf{e}_2$ can then be derived from $\mathbf{e}_1$ and $\mathbf{k}$ at any time.

The principal radii of curvature are reinitialized at material discontinuities, and the orientation of the unit vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ may change. If the unit vector in the direction of the incident ray $\mathbf{n}_\mathrm{i}$ is not parallel to the surface normal $\mathbf{n}_\mathrm{s}$, then it is possible to define a unique tangent plane, called the plane of incidence, that contains the incident, reflected, and refracted rays. The unit vector normal to this plane, $\mathbf{u}_0$, is defined as

$$\mathbf{u}_0 = \frac{\mathbf{n}_i \times \mathbf{n}_s}{|\mathbf{n}_i \times \mathbf{n}_s|}$$

If the ray is normal to the surface, then the incident ray is parallel to the refracted ray and antiparallel to the reflected ray, and $\mathbf{u}_0$ can be any arbitrary unit vector orthogonal to $\mathbf{n}_\mathrm{i}$. In addition to the unit vector normal to the plane of incidence, the following unit vectors tangent to the incident wavefront, refracted wavefront, reflected wavefront, and surface normal are defined:

$$\begin{aligned}\mathbf{u}_i &= \mathbf{n}_i \times \mathbf{u}_0\\ \mathbf{u}_t &= \mathbf{n}_t \times \mathbf{u}_0\\ \mathbf{u}_r &= \mathbf{n}_r \times \mathbf{u}_0\\ \mathbf{u}_s &= \mathbf{n}_s \times \mathbf{u}_0\end{aligned}$$

where the subscripts i, t, and r denote the incident, refracted, and reflected wavefronts, respectively. For a wavefront propagating in a direction $\mathbf{n}$, with principal curvatures $k_1 = 1/r_1$ and $k_2 = 1/r_2$ defined for directions $\mathbf{e}_1$ and $\mathbf{e}_2$, respectively, the principal curvatures in two other orthogonal directions $\mathbf{e}_1'$ and $\mathbf{e}_2'$ (both orthogonal to $\mathbf{n}$) are

$$\begin{aligned}k_1' &= k_1\cos^2\theta + k_2\sin^2\theta\\ k_2' &= k_1\sin^2\theta + k_2\cos^2\theta\\ k_{12}' &= (k_1 - k_2)\cos\theta\sin\theta\end{aligned} \tag{3-1}$$

where θ is the angle of rotation about $\mathbf{n}$ which transforms the vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ to $\mathbf{e}_1'$ and $\mathbf{e}_2'$, respectively. Because k_1' and k_2' are not principal curvatures, it is necessary to include off-diagonal elements of the wavefront curvature tensor equal to k_{12}'.

The following algorithm is used to reinitialize the principal radii of curvature of the wavefront and their orientations. The reinitialization of the curvature variables follows the method of Stavroudis in Ref. 7.

1. Given $\mathbf{n}_i$ and $\mathbf{n}_s$, compute unit vectors in the directions of the reflected and refracted rays, $\mathbf{n}_r$ and $\mathbf{n}_t$.
2. Compute the vectors $\mathbf{u}_0$, $\mathbf{u}_i$, $\mathbf{u}_t$, $\mathbf{u}_r$, and $\mathbf{u}_s$.
3. Compute the angle of rotation $\theta^{(i)}$ needed to transform the local coordinate system with axes parallel to $\mathbf{e}_1$, $\mathbf{e}_2$, and $\mathbf{n}_i$ to a local coordinate system with axes parallel to $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$. (Note: the superscript $^{(i)}$ is used to distinguish this coordinate system rotation angle from the angle of incidence of the ray, often denoted θ_i.)
4. Transform the principal curvatures to the local coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$ by using Equation 3-1, substituting $\theta^{(i)}$ for θ.
5. Given the two principal curvatures of the surface, $k_{1,s}$ and $k_{2,s}$ with directions $\mathbf{e}_{1,s}$ and $\mathbf{e}_{2,s}$, transform the curvature variables to a local coordinate system with axes

parallel to $\mathbf{u}_0$, $\mathbf{u}_s$, and $\mathbf{n}_s$. Let the new curvature variables be denoted by $k_{1,s}'$, $k_{2,s}'$, and $k_{12,s}'$.

6 The curvatures of the wavefront and the surface have now been defined in coordinate systems that share the axis $\mathbf{u}_0$ and only differ by a rotation by the angle of incidence θ_i about $\mathbf{u}_0$. Defining the variables η and γ as in Material Discontinuity Theory, compute the curvature variables of the refracted ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ using the equations

$$k_{1,t}' = \eta k_1' + \gamma k_{1,s}'$$

$$k_{12,t}' = \frac{\eta\cos\theta_i}{\cos\theta_t}k_{12}' + \frac{\gamma}{\cos\theta_t}k_{12,s}'$$

$$k_{2,t}' = \frac{\eta\cos^2\theta_i}{\cos^2\theta_t}k_2' + \frac{\gamma}{\cos^2\theta_t}k_{2,s}'$$

7 Obtain the principal curvatures of the refracted ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ by an angle $\theta^{(t)}$ about $\mathbf{n}_t$. The angle $\theta^{(t)}$ is defined as

$$\theta^{(t)} = \frac{1}{2}\mathrm{atan}\left(\frac{2k_{12,t}'}{k_{1,t}' - k_{2,t}'}\right)$$

8 Reinitialize the principal curvatures of the refracted ray:

$$k_{1,t} = k_{1,t}'\cos^2\theta^{(t)} + 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\sin^2\theta^{(t)}$$
$$k_{2,t} = k_{1,t}'\sin^2\theta^{(t)} - 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\cos^2\theta^{(t)}$$

9 Invert the principal curvatures to obtain the principal radii of curvature of the refracted ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_t$ by the angle $\theta^{(t)}$ to obtain the reinitialized principal curvature direction $\mathbf{e}_{1,t}$.

10 If a reflected ray is released, compute the curvature variables of the reflected ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ using the equations

$$k_{1,r}' = k_1' - 2k_{1,s}'\cos\theta_i$$

$$k_{12,r}' = -k_{12}' + 2k_{12,s}'$$

$$k_{2,r}' = k_2' - \frac{2}{\cos\theta_i}k_{2,s}'$$

11 Obtain the principal curvatures of the reflected ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ by an angle $\theta^{(r)}$ about $\mathbf{n}_r$. The angle $\theta^{(r)}$ is defined as

$$\theta^{(r)} = \frac{1}{2}\operatorname{atan}\left(\frac{2k_{12,r}'}{k_{2,r}' - k_{1,r}'}\right)$$

12 Initialize the principal curvatures of the reflected ray:

$$k_{1,r} = k_{1,r}'\cos^2\theta^{(r)} + 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\sin^2\theta^{(r)}$$
$$k_{2,r} = k_{1,r}'\sin^2\theta^{(r)} - 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\cos^2\theta^{(r)}$$

13 Invert the principal curvatures to obtain the principal radii of curvature of the reflected ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_r$ by the angle $\theta^{(r)}$ to obtain the initialized principal curvature direction $\mathbf{e}_{1,r}$.

In 2D components, only one principal radius of curvature is computed; each ray is treated as a cylindrical wave. In addition, no auxiliary degrees of freedom for the principal curvature direction are required because the out-of-plane direction can always be treated as one of the axes of the local coordinate system that defines the orientation of the wavefront.

In 2D axisymmetric model components, radii of curvature are computed for the in-plane direction, i.e. the *rz*-plane, as well as the out-of-plane or azimuthal directions. During ray-boundary interactions, these radii of curvature are reinitialized as if the ray interacted with a 3D surface of revolution, which may have finite radii of curvature in both the in-plane and out-of-plane directions.

STOKES VECTOR CALCULATION

The values of the intensity I and normalized Stokes parameters are stored as the auxiliary dependent variables I_0, s_{n1}, s_{n2}, and s_{n3} when a ray is released. These auxiliary variables are updated when the ray hits a boundary. At any point along the ray's trajectory, each Stokes parameter is equal to

$$s_0 = I = I_0\frac{r_{1,i}r_{2,i}}{r_1r_2} \qquad s_1 = Is_{n1} \qquad s_2 = Is_{n2} \qquad s_3 = Is_{n3}$$

In 2D, the second principal radius of curvature r_2 is treated as an arbitrarily large distance that remains constant for each ray. Because s_1 and s_2 are based on the differences in intensity between orthogonal polarizations of radiation, a local coordinate system must be defined for the ray. Because auxiliary degrees of freedom have already been allocated for $\mathbf{e}_1$ as explained in Principal Radii of Curvature, a local coordinate system is defined with axes parallel to $\mathbf{e}_1$, $\mathbf{e}_2$, and $\mathbf{k}$. The axes parallel to $\mathbf{e}_1$ and $\mathbf{e}_2$ then function as the x- and y-axes in The Stokes Parameters, respectively.

STOKES VECTOR REINITIALIZATION

Reflection and Refraction of S- and P-polarized Rays

When a fully polarized ray arrives at a material discontinuity, the intensity of the reflected and refracted rays can be computed using the Fresnel equations:

$$\begin{aligned}
t_p &= \frac{2n_1\cos\theta_i}{n_2\cos\theta_i + n_1\cos\theta_t} \\
t_s &= \frac{2n_1\cos\theta_i}{n_1\cos\theta_i + n_2\cos\theta_t} \\
r_p &= \frac{n_2\cos\theta_i - n_1\cos\theta_t}{n_2\cos\theta_i + n_1\cos\theta_t} \\
r_s &= \frac{n_1\cos\theta_i - n_2\cos\theta_t}{n_1\cos\theta_i + n_2\cos\theta_t}
\end{aligned}$$

where the subscripts s and p denote s- and p-polarized rays, or rays with linear polarizations perpendicular to and parallel to the plane of incidence, respectively. The incident ray is assumed to move from a region of refractive index n_1 toward a region of refractive index n_2. The angles θ_i and θ_t are the angle of incidence and angle of refraction, respectively.

If $n_1 > n_2$ and the angle of incidence exceeds the critical angle $\theta = \mathrm{asin}(n_2/n_1)$, the incident ray undergoes total internal reflection, and the reflected ray has intensity equal to that of the incident ray.

The intensity of the transmitted and reflected waves are related to the intensity of the incident wave I_i by the equations

$$\begin{aligned}
I_r &= I_i|r_p|^2 \\
I_t &= \frac{n_2\cos\theta_t}{n_1\cos\theta_i}I_i|t_p|^2
\end{aligned}$$

for p-polarized rays, and

$$I_r = I_i|r_s|^2$$

$$I_t = \frac{n_2\cos\theta_t}{n_1\cos\theta_i}I_i|t_s|^2$$

for s-polarized rays. The coefficients being multiplied by the incident ray intensity are called the *reflectance* R and *transmittance* T:

$$R_p = |r_p|^2$$

$$T_p = \frac{n_2\cos\theta_t}{n_1\cos\theta_i}|t_p|^2$$

$$R_s = |r_s|^2$$

$$T_s = \frac{n_2\cos\theta_t}{n_1\cos\theta_i}|t_s|^2$$

Phase Shift Calculation for Reflected and Refracted Rays

Any fully polarized ray can be resolved into a pair of s- and p-polarized rays, with a phase shift δ between them. The reflected s- and p-polarized rays at a material discontinuity may then undergo different phase shifts; this may, for example, cause a linearly polarized incident ray to be yield a reflected ray with elliptical polarization. In addition, a phase shift may be applied to the transmitted ray if one or more Thin Dielectric Film subnodes are added to a Material Discontinuity node. The phase shifts of s- and p-polarized rays are computed from the complex-valued Fresnel coefficients:

$$\Psi_{s,r} = \arg(r_s)$$
$$\Psi_{p,r} = \arg(r_p)$$
$$\Psi_{s,t} = \arg(t_s)$$
$$\Psi_{p,t} = \arg(t_p)$$

Unpolarized and Partially Polarized Rays

An unpolarized ray has the following properties:

- The instantaneous intensity of the ray changes at random.
- At a given instant in time, the ray has an equal probability of being polarized in any direction orthogonal to the ray trajectory.
- The previous two properties are not affected by the transmission of the ray through a wave retarder or a polarization rotator.

An unpolarized ray has Stokes parameters $s_1 = s_2 = s_3 = 0$.

A partially polarized ray is neither completely deterministic nor completely random. It is characterized by a degree of polarization between 0 and 1.

Often, the ray being reflected or refracted at a boundary is unpolarized or partially polarized; or the polarization direction neither lies in the plane of incidence nor perpendicular to it. In this case, a more general formulation for the new values of the Stokes parameters is needed. The Stokes parameters of the reflected and refracted rays can be expressed in terms of the Stokes parameters of the incident ray, the transmittance or reflectance at the boundary, and the phase delay introduced by the boundary (if any) (Ref. 11):

$$\begin{bmatrix} s_{0t} \\ s_{1t} \\ s_{2t} \\ s_{3t} \end{bmatrix} = \begin{bmatrix} T_s+T_p & T_s-T_p & 0 & 0 \\ T_s-T_p & T_s+T_p & 0 & 0 \\ 0 & 0 & 2\sqrt{T_sT_p}\cos\delta & 2\sqrt{T_sT_p}\sin\delta \\ 0 & 0 & -2\sqrt{T_sT_p}\sin\delta & 2\sqrt{T_sT_p}\cos\delta \end{bmatrix} \begin{bmatrix} s_0 \\ s_1 \\ s_2 \\ s_3 \end{bmatrix}$$

$$\begin{bmatrix} s_{0r} \\ s_{1r} \\ s_{2r} \\ s_{3r} \end{bmatrix} = \begin{bmatrix} R_s+R_p & R_s-R_p & 0 & 0 \\ R_s-R_p & R_s+R_p & 0 & 0 \\ 0 & 0 & -2\sqrt{R_sR_p}\cos\delta & -2\sqrt{R_sR_p}\sin\delta \\ 0 & 0 & 2\sqrt{R_sR_p}\sin\delta & -2\sqrt{R_sR_p}\cos\delta \end{bmatrix} \begin{bmatrix} s_0 \\ s_1 \\ s_2 \\ s_3 \end{bmatrix}$$

where the subscripts *r* and *t* represent reflection and refraction, respectively; and the subscripts *s* and *p* indicate the transmittance or reflectance for s- and p-polarized rays.

Wavefront Curvature Calculation in Graded Media

It is possible to compute the ray intensity by changing **Intensity computation** to **Compute intensity** or **Compute intensity in graded media** in the settings window for the Geometrical Optics interface. The options **Compute intensity and power** and **Compute intensity and power in graded media** can also be used to compute intensity, with the only difference being that these options define an additional auxiliary dependent variable for the total power transferred by the ray. The setting **Compute intensity** is more robust and accurate than **Compute intensity in graded media**, but is only applicable to homogeneous media. The setting **Compute intensity in graded media** can be used for both homogeneous and graded media, but it introduces more numerical error than **Compute intensity**.

ASSUMPTIONS FOR COMPUTING INTENSITY IN HOMOGENEOUS MEDIA

When **Compute intensity** is selected, the ray intensity is computed along each ray path using the following assumptions:

1. The wavefront represented by each ray subtends a constant plane angle (2D) or solid angle (3D), which can change only when rays are reflected or refracted at boundaries.
2. Given the radii of curvature at two positions within the same domain and the intensity at one of these two positions, the intensity at the other position can be computed using the relationship

$$I = I_0 \frac{r_{1,0} r_{2,0}}{r_1 r_2}$$

3. Except when rays are reflected or refracted at boundaries, the principal radii of curvature change at a constant rate:

$$\frac{dr_1}{ds} = \frac{dr_2}{ds} = -1$$

4. The principal curvature directions do not change, except when rays are reflected or refracted at boundaries.

These assumptions are not valid in graded media, in which the refractive index changes continuously as a function of position. It is possible for the angle between two different rays to change as they propagate through the medium, so the solid angle subtended by the wavefront is no longer constant. As a result, the ray intensity cannot be expressed using the ratio of initial and final principal radii of curvature.

CURVATURE TENSOR DEFINITION IN GRADED MEDIA

The calculation of ray intensity in graded media is based on the concept of a curvature tensor $\mathbf{K}$, defined in terms of the principal curvatures κ_1 and κ_2 and the corresponding principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$:

$$\mathbf{K} = \kappa_1 \mathbf{e}_1 \mathbf{e}_1^T + \kappa_2 \mathbf{e}_2 \mathbf{e}_2^T$$

Because $\mathbf{e}_1$ and $\mathbf{e}_2$ are orthogonal, it follows that κ_1 and κ_2 are eigenvalues of $\mathbf{K}$. It also follows that $\mathbf{K}$ is singular because there is no contribution that is orthogonal to both $\mathbf{e}_1$ and $\mathbf{e}_2$.

The signs of the principal curvatures are chosen so that positive curvature indicates that the wavefront is converging, whereas negative curvature indicates that the wavefront is diverging.

The selection of the coordinate system in which the curvature tensor is defined is crucial. It is convenient to describe the coordinate system so that two of the coordinate axes lie in the plane containing $\mathbf{e}_1$ and $\mathbf{e}_2$, since this reduces the number of nonzero terms in $\mathbf{K}$. Further reduction in the number of nonzero terms can be achieved if $\mathbf{K}$ can be defined in a coordinate system in which $\mathbf{e}_1$ and $\mathbf{e}_2$ are basis vectors. This is possible in 2D because one of the principal curvature directions is always parallel to the out-of-plane direction, but in 3D it is not feasible because the principal curvature directions can change as rays propagate through a graded medium.

In the most general 3D case, the curvature tensor is described using the following symbols, each of which corresponds to a different orthonormal basis:

- $\mathbf{K}_X$: curvature tensor defined in the Cartesian coordinate system with basis vectors $\mathbf{x}$, $\mathbf{y}$, and $\mathbf{z}$.
- $\mathbf{K}_W$: curvature tensor defined in the coordinate system consisting of the two principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$ and the direction of propagation $\mathbf{t}$.
- $\mathbf{K}_S$: curvature tensor defined in a coordinate system in which one basis vector is the direction of propagation $\mathbf{t}$. The other two basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ can be determined arbitrarily, as long as they are orthogonal to each other and to $\mathbf{t}$, such that $\mathbf{s}_1 \times \mathbf{s}_2 = \mathbf{t}$. It is more convenient to operate in this coordinate system if the basis vectors can be expressed strictly in terms of $\mathbf{x}$, $\mathbf{y}$, $\mathbf{z}$, and $\mathbf{t}$.

The basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ are defined as follows:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{z}}{|\mathbf{t} \times \mathbf{z}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

To avoid poles in the definition of the basis vectors, the following alternative definitions are used when the rays propagate nearly parallel to the z-axis:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{x}}{|\mathbf{t} \times \mathbf{x}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

The relationship between $\mathbf{s}_1$, $\mathbf{s}_2$, $\mathbf{e}_1$, and $\mathbf{e}_2$ is given by

$$\mathbf{e}_1 = \mathbf{s}_1\cos(\phi) - \mathbf{s}_2\sin(\phi)$$

$$\mathbf{e}_2 = \mathbf{s}_1\sin(\phi) + \mathbf{s}_2\cos(\phi)$$

where the rotation angle ϕ is an auxiliary dependent variable that is stored for each ray.

The relationship between $\mathbf{K}_S$ and $\mathbf{K}_W$ is given by

$$\mathbf{K}_S = \mathbf{Q}_0\mathbf{K}_W\mathbf{Q}_0^T$$

where $\mathbf{Q}_0$ is the rotation matrix:

$$\mathbf{Q}_0 = \begin{bmatrix} \mathbf{s}_1\cdot\mathbf{e}_1 & \mathbf{s}_1\cdot\mathbf{e}_2 & \mathbf{s}_1\cdot\mathbf{t} \\ \mathbf{s}_2\cdot\mathbf{e}_1 & \mathbf{s}_2\cdot\mathbf{e}_2 & \mathbf{s}_2\cdot\mathbf{t} \\ \mathbf{t}\cdot\mathbf{e}_1 & \mathbf{t}\cdot\mathbf{e}_2 & \mathbf{t}\cdot\mathbf{t} \end{bmatrix} = \begin{bmatrix} \cos(\phi) & \sin(\phi) & 0 \\ -\sin(\phi) & \cos(\phi) & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

The relationship between $\mathbf{K}_X$ and $\mathbf{K}_S$ is given by

$$\mathbf{K}_X = \mathbf{Q}\mathbf{K}_S\mathbf{Q}^T$$

where $\mathbf{Q}$ is the rotation matrix:

$$\mathbf{Q} = \begin{bmatrix} \mathbf{x}\cdot\mathbf{s}_1 & \mathbf{x}\cdot\mathbf{s}_2 & \mathbf{x}\cdot\mathbf{t} \\ \mathbf{y}\cdot\mathbf{s}_1 & \mathbf{y}\cdot\mathbf{s}_2 & \mathbf{y}\cdot\mathbf{t} \\ \mathbf{z}\cdot\mathbf{s}_1 & \mathbf{z}\cdot\mathbf{s}_2 & \mathbf{z}\cdot\mathbf{t} \end{bmatrix}$$

In 2D, one of the principal curvature directions is always known, so it is possible to define $\mathbf{s}_1$ and $\mathbf{s}_2$ so that they coincide with $\mathbf{e}_1$ and $\mathbf{e}_{2.}$ The rotation matrix $\mathbf{Q}_0$ is the identity, and the auxiliary dependent variable for the angle ϕ can be omitted.

DERIVATIVES OF THE CURVATURE TENSOR

Following Ref. 9, the derivative of the curvature tensor along the ray trajectory in a graded medium is given by the expression

$$\frac{d\mathbf{K}_S}{ds} = \sum_{i=1}^{5}\mathbf{M}_i$$

where the terms $\mathbf{M}_i$ correspond to the different ways in which the curvature can depend on the medium properties. They are defined using the following expressions:

$$\begin{aligned}\mathbf{M}_1 &= \mathbf{K}_S \cdot \mathbf{K}_S \\ \mathbf{M}_2 &= -\frac{1}{n}(\mathbf{t} \cdot \nabla_S n)\mathbf{K}_S \\ \mathbf{M}_3 &= -\frac{1}{n}\Pi(\nabla_S \otimes \nabla_S n) \\ \mathbf{M}_4 &= \frac{2}{n^2}\Pi(\nabla_S n \otimes \nabla_S n)\Pi \\ \mathbf{M}_5 &= -\frac{1}{n}[(\mathbf{K}\nabla_S n) \otimes \mathbf{t} + \mathbf{t} \otimes \nabla_S n\mathbf{K}]\end{aligned}$$

where Π is the projection matrix:

$$\Pi = \mathbf{I} - (\mathbf{t} \otimes \mathbf{t})$$

The gradient operator ∇_S consists of derivatives that are taken with respect to the local coordinates with basis vectors $\mathbf{s}_1$, $\mathbf{s}_2$, and $\mathbf{t}$.

DERIVATIVES OF THE PRINCIPAL CURVATURES

By application of the chain rule, the derivatives of the nonzero elements of $\mathbf{K}_S$ can be expressed in terms of the principal curvatures that occupy the diagonal elements of $\mathbf{K}_W$ and the rotation angle ϕ :

$$\frac{\partial\kappa_1}{\partial s} = \frac{\partial K_{S,11}}{\partial s}\cos(\phi)^2 - 2\frac{\partial K_{S,12}}{\partial s}\sin(\phi)\cos(\phi) + \frac{\partial K_{S,22}}{\partial s}\sin(\phi)^2$$

$$\frac{\partial\kappa_2}{\partial s} = \frac{\partial K_{S,11}}{\partial s}\sin(\phi)^2 + 2\frac{\partial K_{S,12}}{\partial s}\sin(\phi)\cos(\phi) + \frac{\partial K_{S,22}}{\partial s}\cos(\phi)^2$$

$$\frac{\partial\phi}{\partial s} = \frac{1}{\kappa_2 - \kappa_1}\left[\frac{\partial K_{S,12}}{\partial s}\cos(2\phi) + \frac{1}{2}\left(\frac{\partial K_{S,11}}{\partial s} - \frac{\partial K_{S,22}}{\partial s}\right)\sin(2\phi)\right]$$

NUMERICAL STABILIZATION

The principal curvatures κ_1 and κ_2 are not ideal choices for the auxiliary dependent variables stored by each ray because their values can become arbitrarily large as rays approach caustics. Similarly, the principal radii of curvature r_1 and r_2 are not ideal choices because their values can become arbitrarily large when a diverging wavefront begins to converge while propagating through a graded medium.

Instead, the auxiliary dependent variables stored by each ray are the help variables α_1 and α_2. In 2D, only one help variable is allocated because the out-of-plane principal radius of curvature is assumed to be infinite. The principal curvatures are expressed in terms of the help variables using the expression

$$\kappa_i = \kappa_0 \tan\alpha$$

where $\kappa_0 = 1\ 1/\mathrm{m}$ and $i \in [1, 2]$. The derivatives are then related by the expression

$$\frac{d\alpha_i}{ds} = \frac{1}{\kappa_0}\frac{d\kappa_i}{ds}\cos^2\alpha_i$$

Similarly, the intensity may become infinitely large at caustics, and its reciprocal becomes infinitely large as rays undergo attenuation, so a help variable Γ is used to represent the intensity of the ray:

$$I = I_0 \tan\Gamma$$

where $I_0 = 1\ \mathrm{W/m^2}$. The relationship between the ray intensity help variable and the principal curvatures is

$$\frac{d\Gamma}{ds} = \frac{1}{2}\sin(2\Gamma)[\kappa_1 + \kappa_2 - 2\kappa k_0]$$

where k_0 is the free space wave number and κ is the imaginary part of the refractive index. Three additional auxiliary dependent variables are used to store information about the remaining Stokes parameters.

Refraction in Strongly Absorbing Media

The direction of ray propagation may be described via the loci of points having constant phase or amplitude. In a non-absorbing medium, one which may be described by a real-valued refractive index, changes in the ray intensity are caused only by the convergence or divergence of the wavefront, so the surfaces in which the phase is constant are parallel to the surfaces in which the amplitude is constant.

In a weakly absorbing medium, in which the attenuation of the power transmitted by each ray decays over a length scale much larger than the wavelength, it may be assumed that the surfaces of constant amplitude are still parallel to the surfaces of constant phase. The directions of the reflected and refracted rays at material discontinuities, as well as the Fresnel coefficients that govern their intensity, may be computed using the real parts of the refractive indices on either side of the boundary. However, as the imaginary part of the refractive index increases in magnitude, this assumption may begin to present a considerable source of error because the surfaces of constant amplitude and surfaces of constant phase are no longer parallel.

In the Settings window for the Geometrical Optics interface, select the **Use corrections for strongly absorbing media** check box in the physics interface **Intensity Computation** section to modify Snell's law and the Fresnel equations to accurately model refraction between media with complex refractive indices. This check box is available when **Intensity computation** is set to **Compute intensity** or **Compute intensity and power**.

Following Chang et al. in Ref. 10, the wave vector in an absorbing medium is treated as a bivector with complex components,

$$\mathbf{k} = k_0(N\mathbf{e} - iK\mathbf{f})$$

where $\mathbf{k}$ is the wave vector, k_0 is the wave number in free space, and $\mathbf{e}$ and $\mathbf{f}$ are unit vectors with real-valued components indicating the normal direction to the surfaces of constant phase and surfaces of constant amplitude, respectively. The real-valued quantities N and K, sometimes called the apparent refractive indices, are related to the complex refractive index by the relations

$$N^2 - K^2 = n^2 - \kappa^2$$

$$NK\mathbf{e} \cdot \mathbf{f} = n\kappa$$

where $n - i\kappa$ is the complex refractive index of the medium. From these relations the apparent refractive indices can be computed as long as the dot product $\mathbf{e} \cdot \mathbf{f}$ is known. To store information about the value of this dot product, auxiliary dependent variables for the components of $\mathbf{f}$ are stored when the **Use corrections for strongly absorbing media** check box is selected.

At material discontinuities, the normal vectors to the surfaces of constant amplitude and phase follow a modified form of Snell's law that uses the apparent refractive indices,

$$N_1 \sin\theta_1 = N_2 \sin\theta_2$$

$$K_1 \sin\Psi_1 = K_2 \sin\psi_2$$

where θ and ψ are the acute angles between the surface normal and the normal vectors to the surfaces of constant phase and surfaces of constant amplitude, respectively. The real part of the apparent refractive index in the second domain is a root of the quartic equation

$$AN_2^4 + BN_2^2 + C = 0$$

$$\begin{aligned} A &= 1 \\ B &= -[N_s^2 + K_s^2 + (n_2^2 - \kappa_2^2)] \\ C &= N_s^2 K_s^2 + (n_2^2 - \kappa_2^2)N_2^2 - (nk - N_s K_s \cos\phi)^2 \end{aligned}$$

where $N_s = N_1 \sin\,\theta_1, K_\mathrm{s} = K_1 \sin\,\psi_1$, and the angle ϕ is the azimuthal angle between the normal to surfaces of constant amplitude and the plane of incidence. This equation is obviously quadratic in N_2^2 and can thus be solved using the quadratic formula.

The reinitialized unit normal vectors to the surfaces of constant amplitude and phase are

$$\mathbf{e}_2 = \frac{N_1}{N_2}\mathbf{e}_1 + \left(\cos\theta_2 - \frac{N_1}{N_2}\cos\theta_1\right)\mathbf{n}$$

$$\mathbf{f}_2 = \frac{K_1}{K_2}\mathbf{f}_1 + \left(\cos\psi_2 - \frac{K_1}{K_2}\cos\psi_1\right)\mathbf{n}$$

whereas for the reflected ray the reinitialized unit normal vectors are

$$\mathbf{e}_2 = \mathbf{e}_1 - 2\mathbf{n}\cos\theta$$

$$\mathbf{f}_2 = \mathbf{f}_1 - 2\mathbf{n}\cos\psi$$

Attenuation Within Domains

Rays can gradually lose energy as they propagate through a domain in which the refractive index is complex. The convention of the Geometrical Optics interface is that a lossy medium has refractive index given by $n - i\kappa$. Given the electric field $\mathbf{E}_0$ at one location, the electric field of a ray with infinite wavefront radii of curvature as it propagates through a homogeneous, lossy medium is

$$|\mathbf{E}| = |\mathbf{E}_0|\exp\left(-\frac{k_0 KL}{N}\cos\alpha\right)$$

where L is the optical path length between the points at which $\mathbf{E}$ and $\mathbf{E}_0$ are measured and k_0 is the wave number in free space. In weakly absorbing media, $N = n$, $K = \kappa$, and $\alpha = 0$. If the **Use corrections for strongly absorbing media** check box is selected, N is the real part of the apparent refractive index, K is the complex part of the apparent refractive index, and α is the angle between the surfaces of constant amplitude and surfaces of constant phase; both of these concepts are explained in Refraction in Strongly Absorbing Media. The ray intensity and power are both proportional to the square of the electric field norm, so these quantities change according to the relations

$$I = I_0 \exp\left(-\frac{2k_0 KL}{N}\cos\alpha\right)$$
$$P = P_0 \exp\left(-\frac{2k_0 KL}{N}\cos\alpha\right)$$

When computing the ray intensity in absorbing media, the principal radii of curvature of the wavefront must also be considered. The two contributing factors are accounted for by allocating auxiliary dependent variables for the initial Stokes parameters following the most recent reinitialization of the wave vector, which are expressed as solutions to first-order differential equations of the form

$$\frac{ds_{i,0}}{dt} = -2k_0 K c s_{i,0} \tag{3-2}$$

where c is the speed of light in the medium and $s_{i,0}$ is value of one of the Stokes parameters of the ray before the effects of changes in the principal radii of curvature have been applied.

The Geometrical Optics interface defines an additional dependent variable A (dimensionless) for the path integral of the attenuation coefficient,

$$\frac{dI_0}{dt} = 0 \tag{3-3}$$
$$I = I_0 \exp(A)$$
$$\frac{dA}{dt} = -2k_0 K c$$

Despite requiring an extra degree of freedom per ray, the advantage of Equation 3-3 over Equation 3-2 is that the dependent variables in Equation 3-3 all vary linearly within a homogeneous medium. This makes Equation 3-3 much more stable when the solver takes long time steps or optical path length intervals, compared to Equation 3-2, potentially reducing solution time.

Ray Termination Theory

The Ray Termination feature can annihilate rays when their intensity or power decreases below a specified threshold.

If the ray intensity is computed, then the threshold ray intensity I_{th} (SI unit: W/m^2) can be specified. Then the termination criterion is

$$I < I_{\mathrm{th}}$$

If the ray power is computed, then the threshold ray power Q_{th} (SI unit: W) can be specified. Then the termination criterion is

$$Q < Q_{\mathrm{th}}$$

It is also possible to specify termination criteria based on both intensity and power; in this case, the rays terminate if either criterion is satisfied.

Whenever possible, the **Ray Termination** feature attempts to stop rays at the exact instant when the ray intensity or power reaches a specified threshold value. A closed-form analytic solution for the exact stop time is available under the following circumstances:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a non-absorbing medium.

If neither of these special conditions is met, the ray can only terminate if the termination criterion is met at the beginning of a time step taken by the solver; the ray intensity or power at the instant the ray is stopped may be less than the threshold.

PLANAR WAVEFRONTS IN AN ABSORBING MEDIUM

In this special case, the principal wavefront radii of curvature are so large that the ray intensity can only change significantly in an attenuating medium. The exact time t_s (SI unit: s) at which the ray intensity reaches the threshold value is

$$t_s = t_0 + \tau\log\left(\frac{I}{I_{\mathrm{th}}}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- I (SI unit: W/m^2) is the intensity at t_0, and
- I_{th} (SI unit: W/m^2) is the threshold intensity.

If the termination criterion is based on the ray power Q (SI unit: W) instead of ray intensity, the analogous expression for the stop time is

$$t_s = t_0 + \tau\log\left(\frac{Q}{Q_{\mathrm{th}}}\right)$$

The characteristic time for ray attenuation τ (SI unit: s) is

$$\tau = \frac{\lambda_0 N}{4\pi c K}$$

where

- λ_0 (SI unit: m) is the free-space wavelength of the radiation,
- $N - iK$ (dimensionless) is the apparent refractive index of the medium, where N and K are real numbers, and
- $c = 299{,}792{,}458$ m/s is the speed of light in a vacuum.

DIVERGING WAVEFRONTS IN A NON-ABSORBING MEDIUM

In a non-absorbing medium in which the wavefront is non-planar, the ray can only terminate due to the intensity decreasing below the threshold, since the power does not change. The time t_s (SI unit: s) at which the ray intensity reaches the threshold in a non-attenuating, homogeneous medium is

$$t_s = t_0 + \frac{1}{2V_g}\left(r_1 + r_2 + \sqrt{(r_1 - r_2)^2 + 4\frac{I}{I_{\mathrm{th}}}|r_1 r_2|}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- V_g (SI unit: m/s) is the group velocity magnitude,
- r_1 (SI unit: m) is the first principal radius of curvature of the wavefront at t_0,
- r_2 (SI unit: m) is the second principal radius of curvature of the wavefront at t_0,
- I (SI unit: W/m^2) is the intensity at t_0, and
- I_{th} (SI unit: W/m^2) is the threshold intensity.

In 2D models, or in 3D models where the wavefronts are cylindrical (one extremely large radius of curvature and one finite radius of curvature), the stop time is instead

$$t_s = t_0 + \frac{1}{V_g}\left(r_1 + \frac{I}{I_{\mathrm{th}}}|r_1|\right)$$

where r_1 (SI unit: m) is the finite principal radius of curvature of the cylindrical wavefront.

Illuminated Surface Theory

The Illuminated Surface is used to simulate the emission of rays from a surface, under the assumption that the emitted rays are specularly reflected from an external radiation source. It is assumed that the entire selected surface has direct line of sight to the external radiation source; that is, shadowing due to other geometric entities is not taken into account.

If the Incident ray direction vector is a **User defined direction** or based on **Solar position**, the incident rays are treated as plane waves arriving from a distant source, and the intensity of the incident rays I_i is specified directly.

If the **Incident ray direction vector** is computed using a **User defined point source**, the principal radii of curvature of the incident wavefront are set equal to the distance from the source point to the initial ray position,

$$r_1 = r_2 = -|\mathbf{r}_{\mathrm{src}} - \mathbf{q}|$$

where $\mathbf{q}$ is the initial ray position, $\mathbf{r}_{\mathrm{src}}$ is the position vector of the point source, and r_1 and r_2 are the principal radii of curvature of the incident wavefront. In 2D, only one principal radius of curvature is defined. The incident ray intensity is derived from the total source power P_{src},

$$I_i = \frac{P_{\mathrm{src}}}{4\pi|\mathbf{r}_{\mathrm{src}} - \mathbf{q}|}$$

The principal radii of curvature of the reflected rays are then computed using the approach outlined in Principal Radii of Curvature. Because no thin films are present on the surface, the intensity of the reflected ray is equal to the intensity of the incident ray. The effect of specular reflection is to negate the third and fourth Stokes parameters of the ray.

PERTURBATIONS DUE TO LIMB DARKENING AND SURFACE ROUGHNESS

It is often necessary to take the finite size of a radiation source into account when modeling the reflection or refraction of rays at an illuminated surface. This is particularly true in solar energy applications, in which the finite size of the solar disk significantly affects the focusing capability of solar concentrating devices.

To release rays with a distribution of initial directions based on the finite size of the radiation source, select **Create light cones at release points** or **Sample from conical distribution** option from the **Corrections for finite source diameter** check box in the settings window for the Illuminated Surface feature.

Selecting **Create light cones at release points** causes a cone of rays to be released at each release point. Selecting **Sample from conical distribution** causes only a single ray to be released at each point, but each ray's initial direction is sampled form a cone-shaped probability distribution function. The maximum angle between any incident ray and the specified incident ray direction is the user-defined maximum disk angle ψ_m (SI unit: rad).

The options described above are also available in the settings window for the Solar Radiation feature, in which they have the same effects on the initial direction of the solar radiation.

The default value of the maximum disk angle, or the maximum angle between the direction of the incident rays and the specified **Incident ray direction vector**, is $\psi_m = 4.65$ mrad based on the following expressions (see for example Ref. 14):

$$\psi_m = \operatorname{asin}\left(\frac{d_{\mathrm{sun}}}{2L_0}\right)$$

$$d_{\mathrm{sun}} = 1.392 \times 10^9 \mathrm{m}$$

$$L_0 = 1.496 \times 10^{11} \mathrm{m}$$

When the finite size of the sun is taken into account, radiation from the center of the solar disc tends to be greater in intensity than radiation from the edges of the disc, a phenomenon known as limb darkening. If **Empirical power law** is selected from the **Limb darkening model** list in the settings window for the **Illuminated Surface** or **Solar Radiation** feature, the solar radiation is assumed to follow a power law distribution. Given the intensity $I(0)$ of radiation along a line intersecting the center of the sun, the incident solar radiation from any other point on the sun is given by the expression

$$\frac{I(\psi_s)}{I(0)} = \mu^{\alpha(\lambda_0)}$$

where

$$\mu = \cos\psi_s$$

and ψ_s is the angle between a line of sight to the center of the sun and a line from the center of the sun to another visible point on its surface. The exponent α can vary from 0, for which all solar radiation is of uniform intensity, to 1, at which the intensity of solar radiation falls off linearly as a function of radial position on the visible solar disc. Based on Ref. 14 the exponent α is given as a function of the free-space wavelength λ_0:

$$a(\lambda_0) = \begin{cases} -0.023 + 0.292\lambda_0^{-1} & \lambda_0^{-1} \le 2.4\mu\text{m}^{-1} \\ 0.3778 + 0.125\lambda_0^{-1} & 2.4\mu\text{m}^{-1} \le \lambda_0^{-1} \le 2.8\mu\text{m}^{-1} \\ -0.507 + 0.441\lambda_0^{-1} & 2.8\mu\text{m}^{-1} \le \lambda_0^{-1} \end{cases}$$

If **Linear** is selected from the **Limb darkening model** list, the distribution of solar intensity is instead

$$\frac{I(\psi_s)}{I(0)} = \frac{1+\beta\mu}{1+\beta}$$

It is also possible to include the effect of surface roughness when computing the incident ray direction vector at illuminated surfaces by selecting the **Include surface roughness** check box. The value of the **Surface slope error** σ_ϕ is used as the standard deviation in a Rayleigh distribution of perturbation angles about the unperturbed ray direction vector, from which a user-defined number of rays are sampled.

Theory of Mueller Matrices and Optical Components

Optical devices such as polarizers and wave retarders are often used to manipulate the polarization of radiation. These devices can be modeled by multiplying the Stokes vector of the incident ray $\mathbf{s}_i$ by a Mueller matrix $\mathbf{M}$,

$$\mathbf{s} = \mathbf{M}\mathbf{s}_i$$

As explained in The Stokes Parameters, the Stokes vector requires a set of orthogonal coordinate axes to be defined at the location of a propagating ray. The x-axis of this local coordinate system, $\mathbf{e}_1$, is the first principal curvature direction in 3D models and the out-of-plane direction in 2D models.

Similarly, the Mueller matrix of an optical device is defined with respect to a local coordinate system. This system need not coincide with the coordinate system that defines the Stokes vector of the incident ray. Therefore it is usually necessary to apply a rotation to one of these local coordinate systems before applying the Mueller matrix:

$$\mathbf{s} = \mathbf{R}^{-1}\mathbf{M}\mathbf{R}\mathbf{s}_i$$

where $\mathbf{R}$ is the rotation matrix

$$R = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos(2\theta) & \sin(2\theta) & 0 \\ 0 & -\sin(2\theta) & \cos(2\theta) & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

where θ is the angle by which the local coordinate system of the ray must be rotated so that the x-axes of the two local coordinate systems both lie in the plane of incidence and the y-axes are parallel.

Mueller matrices of some common optical devices are tabulated in Ref. 11.

LINEAR POLARIZER THEORY

A Linear Polarizer is an optical device that produces linearly polarized light. An ideal linear polarizer transmits radiation that is polarized in one direction, indicated by the transmission axis **T**, while completely preventing the transmission of radiation polarized in the orthogonal direction.

In a local coordinate system in which the x-axis is parallel to the transmission axis, the Mueller matrix of the ideal linear polarizer is

$$\mathbf{M} = \frac{1}{2}\begin{bmatrix} 1 & 1 & 0 & 0 \\ 1 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

LINEAR WAVE RETARDER THEORY

A Linear Wave Retarder is an optical device that separates a ray into two rays with orthogonal linear polarizations, retards the phase of one of these rays with respect to the other, and then recombines the two rays. The direction of polarization that undergoes this retardation is called the slow axis, and the orthogonal direction is called the fast axis **F**. The phase delay between the two orthogonal components is the retardance δ of the device.

In a local coordinate system in which the x-axis is parallel to the fast axis, the Mueller matrix of an ideal linear wave retarder is

$$\mathbf{M} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & \cos\delta & \sin\delta \\ 0 & 0 & -\sin\delta & \cos\delta \end{bmatrix}$$

For example, a quarter-wave retarder ($\delta = \pi/2$) converts circularly polarized light to linearly polarized light, whereas a half-wave retarder ($\delta = \pi$) converts left-handed circularly polarized light to right-handed circularly polarized light.

CIRCULAR WAVE RETARDER THEORY

A Circular Wave Retarder, or polarization rotator, is an optical device that separates a ray into two rays with left-handed and right-handed circular polarizations, retards the phase of one of these rays with respect to the other, and recombines the two rays. A right circular polarization rotator is represented by the Mueller matrix

$$\mathbf{M} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos\delta & \sin\delta & 0 \\ 0 & -\sin\delta & \cos\delta & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

The circular retarder rotates the polarization ellipse of the incident ray by an angle $\delta/2$ about the origin.

IDEAL DEPOLARIZER THEORY

The Ideal Depolarizer converts radiation of any polarization to completely unpolarized radiation. The Mueller matrix of an ideal depolarizer is

$$\mathbf{M} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Thin Dielectric Film Theory

When ray intensity or power is solved for, thin dielectric films can be used to change the reflection and transmission coefficients at boundaries. Individual dielectric layers can be applied to a boundary by using the Thin Dielectric Film subnode, which can be added to the Material Discontinuity and Wall nodes. The **Thin Dielectric Film** subnode can be added to the same boundary multiple times to create multilayer films. In general, the thickness of these thin dielectric films is comparable in magnitude to the wavelength of the rays; a minimum requirement is that the thickness of each layer is much less than the coherence length of the incident rays.

COMPUTING THE FRESNEL COEFFICIENTS OF BOUNDARIES WITH THIN DIELECTRIC FILMS

When the **Thin Dielectric Film** feature is used to model dielectric layers, the equivalent Fresnel coefficients for ray propagation through the layer are computed using a recursive algorithm. These equivalent Fresnel coefficients are, in general, complex-valued reflection and transmission coefficients that account for the phase delay and amplitude change in the reflected and transmitted rays resulting from the multiple reflections that the ray undergoes within the dielectric layer, all of which are considered to be coherent with each other. As these multiple reflected and refracted rays propagate into the domains adjacent to the film, they can interfere constructively or destructively with each other, which affects the total electric field amplitude that propagates into each domain. This is illustrated in Figure 3-8 below. Due to the rays' capability to interfere constructively or destructively with each other, the optical thickness of the thin film has a significant effect on the amplitude and phase of the reflected and refracted rays.

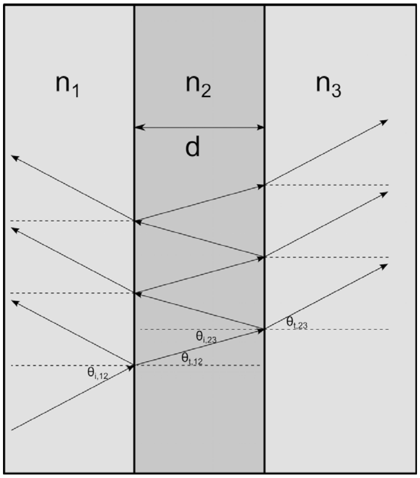


Figure 3-8: One incident ray enters a thin dielectric film and produces a large number of coherent reflected and refracted rays in the adjacent semi-infinite media. These rays can interfere constructively or destructively with each other.

Consider, as shown above, a thin film of thickness d and refractive index n_2 bounded by semi-infinite domains with refractive indices n_1 and n_3. Also consider a linearly polarized ray with electric field amplitude E_0 that propagates through the domain of refractive index n_1 and interacts with the thin film. Let the coefficients of refraction and transmission for a ray moving from the domain of refractive index n_a to the domain of refractive index n_b be denoted r_{ab} and t_{ab}, respectively. When the ray arrives at the film, the refracted ray undergoes a series of reflections at the boundaries between the film and either adjacent domain. Following Ref. 6, the amplitudes of the refracted rays in either domain form a pair of converging geometric series with sums

$$r_{\mathrm{eq}} = \frac{E_1}{E_0} = r_{12} + t_{12}t_{21}r_{23}e^{2i\beta}\sum_{k=0}^{\infty}(r_{21}r_{23}e^{2i\beta})^k = \frac{r_{12}+r_{23}e^{2i\beta}}{1+r_{12}r_{23}e^{2i\beta}}$$

for rays returning to the domain of refractive index n_1, and

$$t_{\mathrm{eq}} = \frac{E_3}{E_0} = t_{12}t_{23}e^{i\beta}\sum_{k=0}^{\infty}(r_{21}r_{23}e^{2i\beta})^k = \frac{t_{12}t_{23}e^{i\beta}}{1+r_{12}r_{23}e^{2i\beta}}$$

for rays entering the domain of refractive index n_3. The angle β is the phase delay that is introduced when a ray propagates from one surface of the film to the other,

$$\beta = \frac{2\pi n_2 d\cos\theta_2}{\lambda_0} \tag{3-4}$$

where λ_0 is the free-space wavelength and θ_2 is the angle between the wave vector of a ray within the thin film and the normal to the film. The dielectric film can be modeled as a material discontinuity with transmission and reflection coefficients $t_{\mathrm{eq}} = E_3 / E_0$ and $r_{\mathrm{eq}} = E_1 / E_0$.

Several derivations for the equivalent Fresnel coefficients are also given by Heavens in Ref. 12, including recursive algorithms and matrix formulations. However, the sign conventions and symbols used in this section have been selected to be consistent with Born and Wolf (Ref. 6) and therefore differ from those in Ref. 12.

AUTOMATIC SETUP OF SINGLE-LAYER FILMS

The Material Discontinuity node includes options for automatic setup of single-layer films with specified reflectance R or transmittance T. In either case the layers are assumed to be non-absorbing, so that $R + T = 1$. To enable these options, select **Specify reflectance** or **Specify transmittance** from the **Thin dielectric films on boundary** list, and then select the **Treat as single layer dielectric film** check box.

The specified reflectance R of a thin film with refractive index n is valid only for certain values of the angle of incidence θ_i and free-space wavelength λ_0. After computing the Fresnel coefficients for rays entering and leaving the layer, the reflectance can be expressed as

$$R = \left|\frac{r_{12} + r_{23}e^{2i\beta}}{1 + r_{12}r_{23}e^{2i\beta}}\right|^2 = \frac{r_{12}^2 + r_{23}^2 + 2r_{12}r_{23}\cos(2\beta)}{r_{12}^2r_{23}^2 + 2r_{12}r_{23}\cos(2\beta) + 1}$$

Solving this expression for β and substituting the result into Equation 3-4 yields

$$d = \left|\frac{\lambda_0}{4\pi n\cos\theta_2}\operatorname{acos}\left(\frac{r_{12}^2r_{23}^2R - r_{12}^2 - r_{23}^2 + R}{2r_{12}^2r_{23}^2(1-R)}\right)\right|$$

For nonzero angles of incidence, the layer thickness that yields a specified reflectance or transmittance depends on the ray polarization. The coefficients of reflection for rays propagating into or out of the layer are either those for S- or P-polarized radiation, depending on the selected option in the **Specified film behavior applies to** list.

⚠ In some cases, the specified transmittance or reflectance will be impossible to achieve, given the refractive indices of the layer and the surrounding media. In such cases, rays will pass through the boundary as if no dielectric layers were present.

AUTOMATIC SETUP OF ANTI-REFLECTIVE COATINGS

To automatically compute the refractive index and thickness of a single-layer anti-reflective coating between two dielectric media, in the settings for the Material Discontinuity node select **Anti-reflective coating** from the **Thin dielectric films on boundary** list. By default the anti-reflective coating is ideal, in the sense that is always has a reflectance of zero for rays of any wavelength and any angle of incidence. However, if you select the **Treat as single layer dielectric film** check box, then instead the single-layer coating only has a reflectance of zero for rays at a specific angle of incidence θ_i (SI unit: rad) and vacuum wavelength λ_0 (SI unit: m).

For oblique incidence, the refractive index of the ideal single-layer coating differs for s- and p-polarized rays. For s-polarized rays, the refractive index n_2 of a film between media with refractive indices n_1 and n_3 is

$$n_2 = \sqrt{(n_1\sin\theta_i)^2 + n_1\cos\theta_i\sqrt{n_3^2 - (n_1\sin\theta_i)^2}}$$

For p-polarized rays, the refractive index is

$$n_2 = \frac{\sqrt{2}}{2}\sqrt{\frac{\cos\theta_i n_a n_1(n_3 + \sqrt{n_3^2 - 4\cos\theta_i \sin^2\theta_i n_a n_1 n_3})}{n_a \cos\theta_i}}$$

where

$$n_a \equiv \frac{n_3^2 - (n_1 \sin\theta_i)^2}{n_3^2}$$

Either equation returns the value of n_2 for which $r_{12} = r_{23}$ for rays of the specified polarization at the specified angle of incidence. An ideal anti-reflective coating is obtained when the magnitude of the equivalent reflection coefficient is 0. This can be obtained when $r_{12} = r_{23}$ and the phase delay δ is equal to $\pi/2$. Substituting this requirement into Equation 3-4 yields an expression for d:

$$d = \frac{\lambda_0}{4n_2\cos\theta_2}$$

SETUP OF REPEATING MULTILAYER FILMS

It is possible to identify a unit cell consisting of a small number of thin dielectric films and to repeat these layers a specified number of times. To set up repeating multilayer films, select **Add layers to surface, repeating** from the **Thin dielectric films on boundary** list.

Select the **Repeat layer in multilayer films** check box in the **Repeating Multilayer Films** section of the Thin Dielectric Film settings window to include the layer in the unit cell of a repeating multilayer film. It is possible to include some layers in the unit cell while excluding others. This could be used, for example, to ensure that a repeating multilayer film begins and ends with the same layer, since in this case the last layer is not included in any unit cell. However, the **Thin Dielectric Film** nodes for all layers in the unit cell must be adjacent to each other in the Model Builder.

Enter a value or expression for the **Number of repeating unit cells** N in the Material Discontinuity settings window. If there are p **Thin Dielectric Film** nodes in which the **Repeat layer in multilayer films** check box is selected and q **Thin Dielectric Film** nodes in which the **Repeat layer in multilayer films** check box is cleared, the total number of dielectric layers in the multilayer film N_t will be $Np + q$.

The recursive algorithm described in the section Computing the Fresnel Coefficients of Boundaries with Thin Dielectric Films is not well-suited to the calculation of the effective Fresnel coefficients for an extremely large number of thin dielectric films in a periodic arrangement. Instead, the following approach is used, in which the equivalent Fresnel coefficients are expressed in terms of the product of a large number of 2-by-2 matrices.

Let the electric field amplitude of the incident, reflected, and transmitted rays entering layer m be denoted E_m^i, E_m^r, and E_{m+1}^i, respectively, where $m \in [1, N_t]$. The ratios of the incoming and outgoing electric field amplitudes in consecutive layers m and m+1 are

$$\begin{bmatrix} E_m^i \\ E_m^r \end{bmatrix} = \frac{1}{t_{(m-1)(m)}} \mathbf{C}_m \begin{bmatrix} E_{m+1}^i \\ E_{m+1}^r \end{bmatrix}$$

Where $\mathbf{C}_m$ is a 2-by-2 matrix:

$$\mathbf{C}_m = \begin{bmatrix} C_{m,11} & C_{m,12} \\ C_{m,21} & C_{m,22} \end{bmatrix} = \begin{bmatrix} \exp(-i\delta_{m-1}) & r_{(m-1)(m)}\exp(-i\delta_{m-1}) \\ r_{(m-1)(m)}\exp(i\delta_{m-1}) & \exp(i\delta_{m-1}) \end{bmatrix}$$

where $\delta_{m\text{-}1}$ is the phase shift within the layer, with $\delta_0 = 0$.

For a multilayer film containing N_t layers including N unit cells, the relationship between the incoming and outgoing electric field amplitudes can be expressed using the product of $N_t + 1$ matrices and $N_t + 1$ transmission coefficients:

$$\begin{bmatrix} E_0^i \\ E_0^r \end{bmatrix} = \left(\prod_{m=1}^{N_t+1} t_{(m-1)(m)} \right)^{-1} \left(\prod_{m=1}^{N_t+1} \mathbf{C}_m \right) \begin{bmatrix} E_{N_t+1}^i \\ E_{N_t+1}^r \end{bmatrix}$$

To simplify the matrix multiplication, the product of the matrices can be expressed in the form

$$\prod_{m=1}^{N_t+1} \mathbf{C}_m = \mathbf{C}_{\mathrm{in}}(\mathbf{C}_{\mathrm{cell}})^N \mathbf{C}_{\mathrm{out}}$$

where $\mathbf{C}_{\mathrm{in}}$ is the product of the matrices for all layers prior to the unit cell, $\mathbf{C}_{\mathrm{out}}$ is the product of the matrices for all layers following the unit cell, and $\mathbf{C}_{\mathrm{cell}}$ is the matrix for all layers within one unit cell, including the interface between one unit cell and the next. Next an eigenvalue decomposition is performed on $\mathbf{C}_{\text{cell}}$:

$$\mathbf{C}_{\text{cell}} = \mathbf{XDX}^{-1}$$

where $\mathbf{D}$ is a diagonal matrix with nonzero elements D_{11} and D_{22}. The ratio of the incoming and outgoing electric field amplitudes can then be expressed as

$$\begin{bmatrix} E_0^i \\ E_0^r \end{bmatrix} = \left(\prod_{m=1}^{N_t+1} t_{(m-1)(m)}\right)^{-1} \mathbf{C}_{\text{in}}\mathbf{X}\begin{bmatrix} D_{11}^N & 0 \\ 0 & D_{22}^N \end{bmatrix}\mathbf{X}^{-1}\mathbf{C}_{\text{out}}\begin{bmatrix} E_{N_t+1}^i \\ E_{N_t+1}^r \end{bmatrix}$$

Because there is no reflected ray beyond the last dielectric layer, $E_{N_t+1}^r = 0$ and the equivalent Fresnel coefficients of the multilayer film can now be obtained:

$$r_{\text{eq}} = \frac{E_0^r}{E_0^i}$$

$$t_{\text{eq}} = \frac{E_{N_t+1}^i}{E_0^i}$$

Grating Theory

The Grating feature releases secondary rays of arbitrary diffraction order. The angles at which transmitted and reflected diffraction orders are released are those at which the waves generated by successive unit cells interfere constructively with each other. A transmitted wave of diffraction order m corresponds to an angle θ_m with the boundary normal so that

$$m\lambda_0 = d(n_2\sin\theta_m - n_1\sin\theta_i)$$

where λ_0 (SI unit: m) is the vacuum wavelength, d (SI unit: m) is the grating constant, θ_i (SI unit: rad) is the angle of incidence, and n_1 and n_2 (dimensionless) are the refractive indices on the incoming and outgoing sides of the boundary, respectively. The corresponding relation for the reflected wave is

$$m\lambda_0 = dn_1(\sin\theta_m - \sin\theta_i)$$

Interference Pattern Theory

The **Interference Pattern** plot can be used to visualize the fringes resulting from the interference of two or more rays at a surface. The interference pattern is only valid when it is plotted over an area with a length scale that is much smaller than the principal radius of curvature of any incident wavefront. This is a consequence of the treatment of each ray as a wavefront that subtends a small solid angle. Furthermore, this plot type requires calculation of the ray intensity and instantaneous phase.

Given the phase Ψ_0 of a ray at the point (q_x, q_y) where the ray intersects a plane, the phase Ψ at a nearby point $(q_x + \Delta x, q_y + \Delta y)$ is computed as follows. Without loss of generality assume that the x- and y-axes are oriented so that they are parallel to the projections of the principal curvature directions onto the cut plane. For a spherical wavefront, the orientations of these axes may be determined arbitrarily as long as they are orthogonal. Let $n_{i,x}$ and $n_{i,y}$ denote the x- and y- components, respectively, of the normalized ray direction vector. Let r_1 and r_2 denote the principal radii of curvature in the directions parallel to the x- and y-axes, respectively. For an incident ray with wave vector magnitude k, the phase at $(q_x + \Delta x, q_y)$ is

$$\Psi(q_x + \Delta x, q_y) = \Psi(q_x, q_y) + k\Delta r_1$$

Where Δr_1 is the change in the principal radius of curvature of the wavefront when moving between the two points.

$$(r_1 + \Delta r_1)^2 = [r_1{}^2 - (n_{i,x}r_1)^2] + (n_{i,x}r_1 + \Delta x)^2$$

Further simplification yields

$$\Delta r_1 = \sqrt{r_1{}^2 + 2n_{i,x}r_1\Delta x + \Delta x^2} - r_1$$

This equation yields an indeterminate form in the limit of infinite radii of curvature, making it unsuitable for computing interference patterns involving plane waves. The equation can be made more robust for plane waves by applying a Taylor series approximation about $\Delta x = 0$,

$$\sqrt{r_1{}^2 + 2n_{i,x}r_1\Delta x + \Delta x^2} = r_1 + n_{i,x}\Delta x + \frac{1}{2r_1}(1 - n_{i,x}^2)\Delta x^2 + O(\Delta x^3)$$

Discarding terms of order Δx^3 or higher yields

$$\Delta r_1 = n_{i,x}\Delta x + \frac{1}{2r_1}(1 - n_{i,x}^2)\Delta x^2$$

Similarly, the change in principal radius of curvature due to a small perturbation in the y-direction is

$$\Delta r_2 = n_{i,y}\Delta y + \frac{1}{2r_2}(1 - n_{i,y}^2)\Delta y^2$$

The rationale for discarding higher-order terms is that they all involve division by higher powers of r_1 and r_2. Therefore this series approximation is only valid when the perturbations in the x and y directions are much smaller than the principal radii of curvature. To assume otherwise would mean that the incident wavefronts subtend very large solid angles.

Combining the perturbations in the two principal radii of curvature yields an expression for the phase at any point $(q_x + \Delta x, q_y + \Delta y)$ on the cut plane,

$$\Psi = \Psi_0 + k\left[n_{i,x}\Delta x + \frac{1}{2r_1}(1 - n_{i,x}^2)\Delta x^2 + n_{i,y}\Delta y + \frac{1}{2r_2}(1 - n_{i,y}^2)\Delta y^2\right]$$

The incident Stokes vector is then converted to an incident electric field using the relation

$$I = \frac{n}{2}\sqrt{\frac{\varepsilon_0}{\mu_0}}(\mathbf{E} \cdot \mathbf{E}^*)$$

and the electric field at the intersection of the ray with the cut plane can be used to compute the electric field at a nearby point using the relation

$$\mathbf{E}(q_x + \Delta x, q_y + \Delta y) = \mathbf{E}(q_x, q_y)\exp[\Psi(q_x + \Delta x, q_y + \Delta y) - \Psi_0]$$

where it is again assumed that the ray subtends a small solid angle so that the intensity of the incident ray is approximately uniform over the region in which the interference pattern is plotted. It is then straightforward to compute the sum of the electric fields of all incident rays and derive the total incident intensity from the electric field norm.

Results Analysis and Visualization in the Ray Optics Modeling chapter.

Monochromatic Aberrations and Zernike Polynomials

A standard way to quantify monochromatic aberrations is to express the optical path difference of all incident rays as a linear combination of Zernike polynomials. With the Ray Optics Module, the Zernike coefficients can be reported in a table using the **Aberration Evaluation** feature. Alternatively, a superposition of different Zernike polynomials can be shown on a unit circle using the **Optical Aberration** plot.

Several different standards exist for naming, normalizing, and organizing the Zernike polynomials. The approach used in this section follows the standards published by the International Organization for Standardization (ISO, Ref. 15) and the American National Standards Institute (ANSI, Ref. 16).

Each Zernike polynomial Z_n^m can be expressed as

$$Z_n^m = N_n^m R_n^{|m|}(\rho) M(m\theta)$$

where

- N_n^m is the normalization term,
- $R_n^{|m|}$ is the radial term,
- $M(m\theta)$ is the meridional term or azimuthal term,
- ρ is the radial parameter, given by $\rho = r/a$ where r is the distance from the aperture center and a is the aperture radius, so that $0 \le \rho \le 1$,
- θ is the meridional parameter or azimuthal angle, $0 \le \theta \le 2\pi$,
- the lower index n is a nonnegative integer, $n = 0,1,2\ldots$, and
- the upper index m is an integer, $m = -n, -n+2, \ldots, n-2, n$ so that $n - |m|$ is always even.

The normalization term N_n^m is

$$N_n^m = (2 - \delta_{0,m})(n+1)$$

where $\delta_{0,m}$ is the Kronecker delta,

$$\delta_{i,j} = \begin{cases} 1 & i = j \\ 0 & i \ne j \end{cases}$$

The radial term is given by the equation

$$R_n^m(\rho) = \sum_{s=0}^{0.5(n-|m|)} \frac{(-1)^s(n-s)!}{s![0.5(n+|m|)-s]![0.5(n-|m|)-s]!}\rho^{n-2s}$$

where “!” denotes the factorial operator; for nonnegative integers,

$$n! = \begin{cases} 1 & n = 0 \\ 1 \times 2 \times 3 \times \ldots \times n & n > 0 \end{cases}$$

The meridional term is given by the equation

$$M(m\theta) = \begin{cases} \cos(m\theta) & m \ge 0 \\ \sin(|m|\theta) & m < 0 \end{cases}$$

The Zernike polynomials thus defined are normalized Zernike polynomials. They are orthogonal in the sense that any pair of Zernike polynomials satisfy the equation

$$\int_0^1 \rho d\rho \int_0^{2\pi} Z_n^m Z_{n'}^{m'} d\theta = \pi\delta_{n,n'}\delta_{m,m'}$$

The normalized Zernike polynomials up to fifth order, along with their common names, are given in Table 3-4.

TABLE 3-4: ZERNIKE POLYNOMIALS

TERM	EXPRESSION	COMMON NAME
Z_0^0	1	Piston
Z_1^{-1}	$2\rho\sin(\theta)$	Vertical tilt
Z_1^1	$2\rho\cos(\theta)$	Horizontal tilt
Z_2^{-2}	$\sqrt{6}\rho^2\sin(2\theta)$	Oblique astigmatism
Z_2^0	$\sqrt{3}(2\rho^2-1)$	Defocus
Z_2^2	$\sqrt{6}\rho^2\cos(2\theta)$	Astigmatism
Z_3^{-3}	$\sqrt{8}\rho^3\sin(3\theta)$	Oblique trefoil

TABLE 3-4: ZERNIKE POLYNOMIALS

TERM	EXPRESSION	COMMON NAME
Z_3^{-1}	$\sqrt{8}(3\rho^3-2\rho)\sin(3\theta)$	Vertical coma
Z_3^1	$\sqrt{8}(3\rho^3-2\rho)\cos(3\theta)$	Horizontal coma
Z_3^3	$\sqrt{8}\rho^3\cos(3\theta)$	Horizontal trefoil
Z_4^{-4}	$\sqrt{10}\rho^4\sin(4\theta)$	Oblique trefoil
Z_4^{-2}	$\sqrt{10}(4\rho^4-3\rho^2)\sin(2\theta)$	Oblique secondary astigmatism
Z_4^0	$\sqrt{5}(6\rho^4-6\rho^2+1)$	Spherical aberration
Z_4^2	$\sqrt{10}(4\rho^4-3\rho^2)\cos(2\theta)$	Secondary astigmatism
Z_4^4	$\sqrt{10}\rho^4\cos(4\theta)$	Horizontal trefoil
Z_5^{-5}	$\sqrt{12}\rho^5\sin(5\theta)$	
Z_5^{-3}	$\sqrt{12}(5\rho^5-4\rho^3)\sin(3\theta)$ †	
Z_5^{-1}	$\sqrt{12}(10\rho^5-12\rho^3+3\rho)\sin(\theta)$ †	
Z_5^1	$\sqrt{12}(10\rho^5-12\rho^3+3\rho)\cos(\theta)$ †	
Z_5^3	$\sqrt{12}(5\rho^5-4\rho^3)\cos(3\theta)$ †	
Z_5^5	$\sqrt{12}\rho^5\cos(5\theta)$	

† This term differs from the expression in Table E.1 in ISO 24157:2008 (Ref. 15), which contains an error.

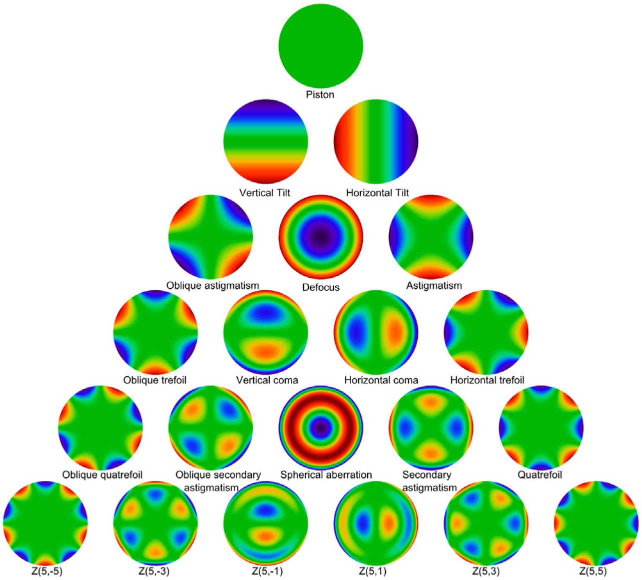


Figure 3-9: Zernike polynomials on the unit circle.

Accumulator Theory: Domains

The Accumulator (Domain) node is used to transfer information from rays to the domains they occupy or pass through. Each **Accumulator** defines a variable, called the accumulated variable, in the selected domains. The accumulated variable is discretized using constant shape functions, so its value is uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** edit field in the **Accumulator Settings** section of the settings window. The default variable name, rpd, will be used in the remainder of this section when referring to the accumulated variable.

ACCUMULATOR TYPE

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the area or volume of the mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the area or volume of the mesh element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpd}_{\mathrm{count}} = \mathrm{rpd}_{\mathrm{density}} \times V$$

where V is the mesh element volume (in 3D) or area (in 2D).

ACCUMULATION OVER ELEMENTS

When **Elements** is selected from the **Accumulate over** list, the value of the accumulated variable in a mesh element is the sum of the source terms R_i evaluated for all rays in that mesh element:

$$\mathrm{rpd} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

where N is the total number of rays in the element and V is the area or volume of the mesh element. In other words, the contribution of each ray to the accumulated variable is distributed uniformly over the mesh element the ray is in, regardless of the ray's exact position within the element.

If **Elements and time** is selected from the **Accumulate over** list, then the sum of the source terms for rays in the mesh element is used to define the time derivative of the accumulated variable, rather than its instantaneous value:

$$\frac{d(\mathrm{rpd})}{dt} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

Thus the value of the accumulated variable depends on the time history of rays in the mesh element, instead of the instantaneous positions of the rays. As each ray propagates, it will leave behind a trail based on its contributions to the accumulated variables in mesh elements it has traversed. The algorithm for accumulating over time takes into account the fraction of a time step taken by the solver that the ray spends in each mesh element, even if it crosses between elements during the time step.

Accumulator Theory: Boundaries

The Accumulator (Boundary) feature transfers information from rays to the boundaries they hit or pass through. Each **Accumulator** defines a variable, called the accumulated variable, on the selected boundaries. The accumulated variable is discretized using constant shape functions, so its value if uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** edit field in the **Accumulator Settings** section of the settings window. The default variable name, rpb, will be used in the remainder of this section when referring to the accumulated variable.

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the surface area or length of the boundary mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the surface area or length of the boundary element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpb}_{\mathrm{count}} = \mathrm{rpb}_{\mathrm{density}} \times V$$

where V is the boundary element surface area (in 3D) or length (in 2D).

When **Rays in boundary elements** is selected from the **Accumulate over** list, the accumulated variable in a boundary element gets incremented by the source term R whenever a ray freezes or sticks to the boundary:

$$\mathrm{rpb}_{\mathrm{new}} = \mathrm{rpb} + \frac{R}{V}$$

where division by the mesh element area or length occurs because the accumulator is assumed to be of type **Density**. Thus the source term evaluated for an incident ray is uniformly distributed over the boundary element it freezes or sticks to.

If instead **Ray-wall interactions** is selected from the **Accumulate over** list, then the accumulated variable gets incremented regardless of what type of ray-wall interaction occurs. Thus, it is possible for the same ray to increment the accumulated variable in many different boundary elements, or even in the same element multiple times.

BUILT-IN GLOBAL VARIABLES

By default, the boundary **Accumulator** defines the following global variables:

TABLE 3-5: BUILT-IN GLOBAL VARIABLES FOR BOUNDARY ACCUMULATORS

NAME	EXPRESSION
`<scope>.<name>_ave`	Average of accumulated variable
`<scope>.<name>_int`	Integral of accumulated variable

TABLE 3-5: BUILT-IN GLOBAL VARIABLES FOR BOUNDARY ACCUMULATORS

NAME	EXPRESSION
`<scope>.<name>_max`	Maximum of accumulated variable
`<scope>.<name>_min`	Minimum of accumulated variable
`<scope>.<name>_sum`	Sum of accumulated variable over elements

Here, `<scope>` includes the physics interface name and the names the Accumulator and parent feature. For example, the average of the accumulated variable over a boundary may be called gop`.wall1.bacc1.rpb_ave`, where `gop` is the name of the Geometrical Optics interface, `wall1` is the name of the parent Wall node, `bacc1` is the name of the Accumulator node, and `rpb` is the accumulated variable name. These variables are all available in the **Add/Replace Expression** menus during results evaluation.

These global variables are computed by defining a set of component couplings on the selection of the parent physics feature, such as the Wall feature to which the Accumulators are added. The following expressions for the global variables are used.

TABLE 3-6: BUILT-IN GLOBAL VARIABLE DEFINITIONS FOR BOUNDARY ACCUMULATORS

NAME	EXPRESSION
`<scope>.<name>_ave`	`<wscope>.aveop(<scope>.<name>)`
`<scope>.<name>_int`	`<wscope>.intop(<scope>.<name>)`
`<scope>.<name>_max`	`<wscope>.maxop(<scope>.<name>)`
`<scope>.<name>_min`	`<wscope>.minop(<scope>.<name>)`
`<scope>.<name>_sum`	`<wscope>.intop(<scope>.<name>/<scope>.meshVol)`

Here, `<wscope>` is the scope of the parent boundary feature, e.g. `goprac.wall1`.

References for the Geometrical Optics Interface

1. L.D. Landau and E.M. Lifshitz, *The Classical Theory of Fields*, 4th ed., Butterworth-Heinemann, Oxford, 1975.

2. B. Edlén, "The dispersion of standard air", J. Optical Society of America, vol. 2, pp. 339-344, 1953.

3. W. J. Tropf, "Temperature-dependent refractive index models for BaF2, CaF2, MgF2, SrF2, LiF, NaF, KCL, ZnS, and ZnSe", Optical Engineering, vol. 34, pp. 1369-1373, 1995.

4. B. J. Frey, D. B. Leviton, T. J. Madison, "Temperature-dependent refractive index of silicon and germanium", *Optomechanical Technologies for Astronomy*, Proc. SPIE, vol. 6273, 2006.

5. H. J. Hoffmann, W. W. Jochs, G. Westenberger, “Dispersion formula for the thermo-optic coefficient of optical glass”, Proc. SPIE, vol. 1327, 1990.

6. M. Born and E. Wolf, *Principles of Optics*, 7th ed., Cambridge, 1999.

7. O.N. Stavroudis, *The Optics of Rays, Wavefronts, and Caustics*, Academic, 1972.

8. B.E.A. Saleh and M.C.Teich, *Fundamentals of Photonics*, Wiley, 1991.

9. R.M. More and K. Kosaka, “Wave-front curvature in geometrical optics”, Phys. Rev. E, vol. 57, no. 5, pp. 6127–6134, 1998.

10. P.C.Y. Chang, J.G. Walker, and K.I. Hopcraft, “Ray tracing in absorbing media”, J. Quantitative Spectroscopy & Radiative Transfer, vol. 96, pp. 327–341, 2005.

11. W. A. Shurcliff, *Polarized Light: Production and Use*, Harvard, 1962.

12. O. S. Heavens, *Optical Properties of Thin Solid Films*, Dover, 1991.

13. H. Anders, *Thin Films in Optics*, Focal Press, 1967.

14. D. Hestroffer and C. Magnan, “Wavelength dependency of the Solar limb darkening”, Astron. Astrophysl, vol. 333, pp. 338–342, 1998.

15. ISO 24157:2008: *Ophthalmic optics and instruments — Reporting aberrations of the human eye*. International Organization for Standardization, Geneva, Switzerland.

16. ANSI Z80.28-2017: *American National Standard for Ophthalmics — Methods of Reporting Optical Aberrations of Eyes*. American National Standards Institute, Alexandria VA.

4

Multiphysics Interfaces and Couplings

The Ray Optics Module contains predefined multiphysics interfaces and couplings to facilitate easy setup of models of ray propagation with significant attenuation and heating. The **Ray Heating** physics interface described in this chapter is found under the **Optics>Ray Optics** branch ( ) when adding a physics interface.

In this chapter:

The Ray Heating Interface

The **Ray Heating** interface ( ) combines the Geometrical Optics interface with the Heat Transfer in Solids interface. The **Ray Heat Source** multiphysics feature is added automatically. The Ray Heating interface is used to model electromagnetic wave propagation in optically large systems in which the refractive index is temperature dependent. The energy lost due to the attenuation of rays in an absorbing medium creates a heat source that is included in the temperature computation.

When a predefined **Ray Heating** interface is added from the **Optics>Ray Optics** branch ( ) of the **Model Wizard** or **Add Physics** windows, **Geometrical Optics** and **Heat Transfer in Solids** interfaces are added to the Model Builder. A **Multiphysics** node is also added, which automatically includes the multiphysics coupling feature **Ray Heat Source**.

On the Constituent Physics Interfaces

The Geometrical Optics interface is used to model electromagnetic wave propagation when the wavelength is much smaller than the smallest geometric entity in the model. It includes built-in tools for computing the intensity, phase, and optical path length of rays. The rays can be polarized, unpolarized, or partially polarized. Frequency distributions and ray propagation in absorbing media are also supported.

The Heat Transfer in Solids interface is used to model heat transfer by conduction, convection, and radiation. A Heat Transfer in Solids model is active by default on all domains and a Heat Transfer in Fluids model is added with an empty selection. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Ray Heating**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Geometrical Optics** and **Heat Transfer in Solids** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose from the available coupling features but the modified settings are not included.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 4-1: MODIFIED SETTINGS FOR A RAY HEATING INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Geometrical Optics	**Compute intensity and power** is selected from the **Intensity computation** list in the physics interface **Ray Properties** section.
Heat Transfer in Solids	No changes.
Ray Heat Source	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Geometrical Optics and Heat Transfer in Solids interfaces are preselected in the **Ray Heat Source** section.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Features

The Ray Heat Source coupling feature node is described in this section.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Geometrical Optics Interface are listed in the section List of Geometrical Optics Interface Physics Features.
- The available physics features for The Heat Transfer Interfaces are listed in the sections Domain Features, Boundary Features, and Edge and Point Features in the *COMSOL Multiphysics Reference Manual*.

Ray Heat Source

Use the **Ray Heat Source** multiphysics coupling ( ) to compute the heat source resulting from the attenuation of rays in an absorbing medium. This heat source is defined to a dependent variable that is defined on all domain mesh elements in the selection of the **Ray Heat Source** node. The ray power should be solved for in the coupled Geometrical Optics interface.

SETTINGS

The **Label** is the default multiphysics coupling name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling in the model) is `rhs1`.

RAY HEAT SOURCE

This section defines the physics involved in the multiphysics coupling. By default, the applicable physics interface is selected in the **Source** and **Destination** lists.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** then the applicable list defaults to **None** as there is nothing to couple to.

If a physics interface is deleted and then added to the model again, and in order to reestablish the coupling, you need to choose the physics interface again from the **Source** or **Destination** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

Theory for the Ray Heating Interface

The Ray Heating Interface theory is described in this section:

- Unidirectional and Bidirectional Couplings
- Coupled Heat Transfer and Ray Tracing Equations
- Heat Source Calculation

Unidirectional and Bidirectional Couplings

The Ray Heating multiphysics interface combines geometrical optics with heat transfer to compute the heat source that is generated as rays propagate through an absorbing medium. The Ray Heating interface can either be used to set up a unidirectional or bidirectional coupling between the constituent physics interfaces.

If the medium properties are not temperature dependent and the domains do not undergo significant thermal expansion, then a unidirectional coupling is appropriate. The heat source generated by the attenuation of rays can be included in the heat transfer simulation by first computing the ray trajectories using a Ray Tracing study step, then computing the temperature using a Stationary study step. Both study steps are described in the *COMSOL Multiphysics Reference Manual*.

If the refractive index of the medium is temperature dependent, a more accurate approach is to set up a bidirectional coupling in which the attenuation of ray power affects the temperature distribution, which in turn affects the ray trajectories. The two physics can be solved for using an iterative solver loop consisting of a time-dependent solver for the ray trajectories and a stationary solver for the temperature. Such a solver sequence can be set up automatically using the Bidirectionally Coupled Ray Tracing study step, also described in the *COMSOL Multiphysics Reference Manual*.

Coupled Heat Transfer and Ray Tracing Equations

Under static conditions, the heat equation can be written as

$$\rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q$$

$$\mathbf{q} = -k \nabla T$$

where

- ρ (SI unit: kg/m^3) is the density,
- C_p (SI unit: J/(kg·K)) is the heat capacity at constant pressure,
- T (SI unit: K) is the temperature,
- Q (SI unit: W/m^3) is the heat source, and
- k (SI unit: W/(m·K)) is the thermal conductivity.

The propagation of each ray can be modeled using a set of coupled first-order equations:

$$\frac{d\mathbf{k}}{dt} = -\frac{\partial\omega}{\partial\mathbf{q}} \qquad \frac{d\mathbf{q}}{dt} = \frac{\partial\omega}{\partial\mathbf{k}}$$

where $\mathbf{k}$ (SI unit: rad/m) is the wave vector, ω (SI unit: rad/s) is the angular frequency, t (SI unit: s) is time, and $\mathbf{q}$ (SI unit: m) is the ray position vector. In an isotropic medium, the frequency and wave vector are related by the expression

$$\omega = \frac{c|\mathbf{k}|}{n(\mathbf{q})}$$

where $c = 2.99792458 \times 10^8$ m/s is the speed of light in a vacuum and n (dimensionless) is the refractive index. If the medium is absorbing, then the intensity of the ray decreases and it deposits some power in the domain as it passes through. This is described in greater detail in the section Attenuation Within Domains in Theory for the Geometrical Optics Interface.

A bidirectional coupling between the Geometrical Optics and Heat Transfer interfaces may be required for any of the following reasons:

- The refractive index is temperature dependent.
- The medium undergoes thermal expansion and the refractive index is strain dependent.
- The medium undergoes thermal expansion, causing rays be reflected or refracted at different angles when entering or leaving the medium.

When none of these conditions is met, the Geometrical Optics interface affects the Heat Transfer interface, and a unidirectional coupling is sufficient.

Heat Source Calculation

Consider a ray j transferring an amount of power Q_j through a domain. During a small time interval Δt, the ray propagates a short distance Δs from initial position $\mathbf{q}_j$ to position $\mathbf{q}_j+\Delta\mathbf{q}_j$ and its power decreases. The deposited ray power in the domain increases by an amount equal in magnitude to the power lost by the ray but opposite in sign:

$$Q_{\mathrm{src}}(\mathbf{r}, t+\Delta t) \approx Q_{\mathrm{src}}(\mathbf{r}, t) - \sum_{j=1}^{N} \frac{\partial Q_j}{\partial t}\Delta t\,\delta(\mathbf{r}-\mathbf{q}_j)$$

where δ is the Dirac delta function. In the limit as this time derivative becomes arbitrarily small, this expression can be rewritten as

$$\frac{dQ_{\mathrm{src}}}{dt} = -\sum_{j=1}^{N} \frac{\partial Q_j}{\partial t}\delta(\mathbf{r}-\mathbf{q}_j)$$

In this expression, every ray is treated as an infinitesimally small point source. This can lead to numerical issues because the heat source becomes infinitely large at the location of each point source. In the following section an alternative expression for the domain heat source is given.

CREATING A BOUNDED HEAT SOURCE TERM

The Ray Heat Source node defines a variable for the contribution to the heat source by rays in each mesh element. This variable is discretized using constant shape functions that are, in general, discontinuous across boundaries between elements. For a mesh element j with volume V_j the average heat source $Q_{\mathrm{src},j}$ changes according to the expression

$$\frac{dQ_{\mathrm{src}}}{dt} = -\frac{1}{V_j}\sum_{j=1}^{N} \frac{\partial Q_j}{\partial t}\int \delta(\mathbf{r}-\mathbf{q}_j)dV$$

The integral on the right-hand side is a volume integral over element j. The resulting time derivative of the heat source is the average value over the mesh element, which may be written more concisely as

$$\frac{dQ_{\mathrm{src},j}}{dt} = -\frac{1}{V_j}\sum_{i=1}^{N_j}\frac{\partial Q_j}{\partial t}$$

where the sum is taken over all rays that are within mesh element j.

5

Glossary

This Glossary of Terms contains modeling terms in a ray optics context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics® software and documentation, see the glossary in the *COMSOL Multiphysics Reference Manual*. For references to more information about a term, see the index.

Glossary of Terms

coherence The degree to which two rays interfere with each other, or to which a ray may interfere with itself when subjected to a time delay.

diffraction order In a diffraction grating, an integer indicating the number of wavelengths between waves emanating from adjacent unit cells before they constructively interfere.

diffuse reflection Reflection at a surface in a random direction, following a probability distribution based on Lambert's cosine law.

Fresnel equations A set of equations for computing the reflection coefficients and transmission coefficients for rays as they cross a boundary between two media with different refractive indices.

geometrical optics The analysis of electromagnetic wave propagation through electrically large domains by treating the electromagnetic waves as rays.

grating A periodic structure from which transmitted and reflected radiation can only propagate in specific directions governed by the frequency of radiation and the size of the unit cell.

intensity A measure of the energy density of an electromagnetic wave; for a plane polarized wave, the intensity is equal to the magnitude of the time-averaged Poynting vector.

marginal ray The ray that is released from the center of the object at the maximum aperture. This ray will pass through the edge of the aperture stop.

monochromatic light Light of uniform frequency or wavelength.

Mueller matrix A 4-by-4 matrix which can be multiplied by the Stokes vector to apply the effects of various optical components to a ray.

optical dispersion model A function giving the refractive index as a function of wavelength. The coefficients for any given model will usually be determined empirically.

optical axis For simply rotationally symmetric optics, this is the axis of rotational symmetry. The optical axis will pass through the center of curvature.

optical path length The integral of refractive index along the ray trajectory. Over a given time interval, the optical path length of a ray is equal to the distance the ray would have propagated in a vacuum.

phase The quantity that indicates the variation of the instantaneous electric field with respect to time and position.

plane of incidence For non-normal incidence of a ray at a surface, the unique plane containing the surface normal and the incident ray direction vector. For normal incidence, any plane containing the surface normal may be selected.

polarization A description of the orientation of the electric field for a polarized or partially polarized ray, and for the way in which this electric field varies as the ray propagates.

polarization, circular The state of polarization in which the electric field amplitude has two orthogonal components of equal magnitude, with a phase shift of $\pi/2$ between them, such that the electric field vector traces a helical pattern as the ray propagates.

polarization, linear The state of polarization in which the electric field only oscillates within a single plane.

power The energy transferred per unit time.

Poynting vector The vector representing the directional energy flux of an electromagnetic field.

primary ray A ray that is released into a model independently of the trajectories of other rays. Primary rays are created by release features.

principal curvature direction A unit vector which, together with the normal to a surface, defines a plane in which one of the principal radii of curvature of a surface is defined.

polychromatic light Light with a distribution of frequencies or wavelengths.

principal curvature The reciprocal of one of the principal radii of curvature of a surface.

principal radii of curvature The maximum and minimum values of the radius of curvature of curves that can be obtained via the intersection of a surface with a plane.

quasi-monochromatic light Light with a distribution of frequencies in which the changes in free-space wavelength are much smaller than the average free-space wavelength.

reflection coefficient A dimensionless coefficient that is used to compute the intensity of reflected rays. At boundaries between media, reflection coefficients are computed using the Fresnel equations.

refractive index The ratio of the speed of light in a vacuum to the speed of light in a medium. This is also known as the absolute refractive index.

refractive index, relative The ratio of the absolute refractive index to the refractive index of air.

release feature A feature that releases a predetermined number of rays and assigns initial values to dependent variables associated with the rays.

secondary ray A ray that is produced by the interaction of existing rays with a domain or boundary feature. The degrees of freedom for all secondary rays are preallocated when beginning to run a study.

Snell's law The set of equations that indicates the directions in which reflected and refracted rays will propagate when a ray interacts with a boundary between two media.

specular reflection Idealized reflection at a smooth surface in which the reflected ray lies within the plane of incidence and the angles of incidence and reflection are equal.

Stokes parameters The four Stokes parameters completely characterize the intensity and polarization of a fully polarized, partially polarized, or unpolarized ray. The Stokes parameters can be interpreted as indicators of the ray intensity that would be measured when sending a ray through various arrangements of polarizers and wave retarders.

thermo-optic dispersion model A function giving the change in refractive index as a function of wavelength and temperature.

transmission coefficient A dimensionless coefficient that is used to compute the intensity of refracted rays at boundaries between media. Transmission coefficients are computed using the Fresnel equations.

wave vector A vector in the direction of ray propagation, which indicates the spatial dependence of the instantaneous electric field for coherent radiation.

wave number The magnitude of the wave vector, equal to ω/c, where ω is the angular frequency and c is the speed of light in the medium.

wave retarder An optical component that applies a phase shift to radiation of a specific polarization with respect to radiation of an orthogonal polarization.

Index