# Table of Contents

**Getting Started** ................................................................. 5  
Cheking the SPEOS CAA V5 Based License ................................ 5  
Setting the Results and Simulations Options ................................. 5  
Multi-Throwing ........................................................................ 6  
Extensions and Units ................................................................... 8  
Changing the User Interface Language .......................................... 11  
Managing Documents .................................................................. 11  
Creating a Set of Features .......................................................... 11  
Using the Other Selection Option .................................................. 11  
OPTIS Distributed Computing Restrictions ..................................... 11  
Monitor Color Calibration ............................................................ 12  
  Understanding the Monitor Display Limitations ............................. 12  
  Why Calibrating a Display ......................................................... 15  
  Doing Color Calibration ............................................................. 16  
  Doing the Characterization ......................................................... 16  
  Matching Color between XMP Results and Calibrated Display ....... 16  
**Features** .................................................................................. 18  
  Optical Properties ...................................................................... 18  
    Face Optical Properties ........................................................... 18  
    Editing the Optical Properties of a Material ............................... 19  
    Ambient Material .................................................................... 20  
    3D Texture ............................................................................ 20  
    LCD Component ...................................................................... 31  
    Polarization Plate Component ................................................ 32  
    Using Rendering Properties as Optical Properties ..................... 34  
  Sources ..................................................................................... 35  
    Surface Source ....................................................................... 35  
    Ambient Sources .................................................................... 40  
    Ray File Source ...................................................................... 49  
    Thermic Surface Source ........................................................... 51  
    Interactive Source ................................................................... 54  
    Luminaire Source ................................................................... 56  
    Creating a Source Group ........................................................... 56  
    Sources Management ............................................................... 57  
    Properties ............................................................................... 58  
  Sensors ...................................................................................... 59  
    Irradiance Sensor ................................................................... 59  
    Intensity Sensor ..................................................................... 73  
    Radiance Sensor .................................................................... 82  
    3D Irradiance Sensor ................................................................ 92  
    3D Energy Density Sensor ....................................................... 93  
    Sensors Management ............................................................... 101  
    Sensors Properties ................................................................. 101  
**Simulations** .......................................................................... 103  
  Simulations Compatibility ........................................................... 103  
  Interactive Simulation .................................................................. 104  
  Direct Simulation ....................................................................... 107  
  Inverse Simulation ..................................................................... 109  
  Virtual BSDF Bench ................................................................... 112  
  Simulations Management ............................................................ 123  
  Simulation Properties ................................................................. 124  
  External Simulations ................................................................... 155  
**Results** .................................................................................. 158  
  Reading the HTML Report .......................................................... 159  
  XMP Result ............................................................................... 159  
  LPF Result ............................................................................... 160  
  Visualizing a Ray Result ............................................................. 163  
  Visualizing an Eulundat, IESNA LM-63, OPTIS Intensity Distribution Result ....................................................... 163  
  Visualizing an Unpolished, Anisotropic BSDF or Complete Scattering Result ....................................................... 163  
  Visualizing a XM3 Result ............................................................. 163  
  Results Management ................................................................. 164  
  Results Properties ................................................................. 165
Checking the SPEOS CAA V5 Based License

You can check and edit the SPEOS CAA V5 Based license directly from the software.

*You must click Tools, Options..., select a package from the tree, and then select the SPEOS Licensing tab.*

- The Server information section displays the name of the server.
  - It is the license server the OPTIS License Manager is connected to.
  - By clicking Connect to Optis portal to request a license, you can make a license request to OPTIS.
- The List of available configurations section displays the available license configurations.
  - You can select a configuration from the list.
  - Configuration represents the list of packages and options.
  - Expiration date informs you about your license expiration date.
  - Version indicates the release number of the OPTIS Software.
  - Tokens acquired represents the number of tokens for in case of a floating license.
  - 0 corresponds to the nodelocked configuration.
  - Tokens in use gives information about the number of tokens used by users.
  - Tx indicates the maximal number of threads.
  - Dx indicates the maximal number of distributed servers.

Setting the Results and Simulations Options

You can configure some behavior of SPEOS CAA V5 Based software concerning result files and simulations.

*You must click Tools, Options..., select a package from the tree to edit the General tab.*

- In the Result section, you can configure results parameters.
  - Automatic launch at end of simulation activates the automatic launch of the results viewer at the end of simulation.
  - Increment name if file already exists activates the automatic increment of the result file name.
    - When it is activated, old result files are not overwritten when a simulation is updated.

| It is not recommended to activate this Increment name if file already exists option. |
| To save your results, it is better to make an isolations. |

- Result concatenator separator activates the choice of the separator automatically added in the result file name.
  - Interactive simulation report impact activates the addition of information related to each impact as position, normal, surface state.
  - Show results in 3D activates the display of results.
  - Show null values of results as transparent in 3D activates the display of results with an advantage of image transparency for null values.

The black color must have been defined as null.
These two last parameters only apply to the next simulations’ results, and not to the results from already created simulations.

To change already created simulations’ results options, you can view Setting the Parameters (see page 165).

- In the Simulation section, you can configure simulations parameters.
  - Thread number defines the number of thread used by direct or inverse simulations.
    You can type a value in the box.
  
  The optimum value for best performance is 2 or 3 for a Hyperthreading processor, or 4 or 5 for a dual processor with hyperthreading.
  
  For more details, you can view Multi-Threading (see page 6).
  - VR Sensor Memory Management activates the memory management.
    Do not unselect the VR Sensor Memory Management box if you do not have enough memory to store entire OptisVR files.
    - Automatic ‘Save All’ before running a Simulation activates the automatic save at simulation launch according to the option parameter.
      This does not apply to interactive simulations neither, because the update is automatic, nor to optimization update cycle.
  
- In the Feature edition section, Automatic alphabetical sort selection list reorders the list of selections by alphabetical order.
  
- In the Display a warning when section, you can configure warnings parameters.
  
  Select the check boxes to configure the warnings’ display when working with the software.
  
  For more details about warnings, you can view Troubleshooting.

Multi-Threading

You must click Tools, Options…, and then select the General tab to configure the thread number.

By default, the Windows applications are monothread.
It means that they use only one processor.

With multi-threading, multiple threads can exist within the context of a single process, sharing the process’ resources but able to execute independently.

The power of processors and computers still continues to grow, as we are reminded by the Moore law:

- Single processors, one physical chip, include the hyper threading technology.
  
  It means that the physical processor is seen as 2 virtual processors.

- Multi-processors: Some computers can include more than 1 processor.
  
  In the past, these computers were dedicated to servers, but now they are becoming increasingly desktop computers.

Windows and the Windows applications can take advantage of this hardware.

When an application can have many virtual or physical processors, it can dispatch a long calculation on all these processors.

The application manages the cooperative access to data to avoid data incoherence.

Performance

Compared to the previous version of SPEOS CAA V5 Based, the performance can be the following on a hyper threading processor:

- 1 thread: Gain between 5% and 15%.
  
  This gain comes from a different management of the progress bar, the periodic saving of maps and the simulation.

- 2 threads: Gain between 20% and 35%.
  
  As it is not really two physical processors and as SPEOS CAA V5 Based manages the cooperative access to data, the gain is lower than 50%.
With a dual processor computer, the gain can be up to 70%.
The gain is more important when simulating complex systems with a lot of geometries.
The gain is very low if the system is only composed of a rectangular source.
If the system to simulate is simple, for the multithreaded simulations, none of the threads ever works at 100% and
adding threads may increase the simulation time (thread management).

Check that Multithreading is Running
When running a multithreading simulation, it is possible to check the use of the processors of SPEOS CAA V5 Based
by using the Windows Task Manager:

Without Multithreading: Number of Threads = 1
One thread is working at 100%.
That is why the CPU Usage is around 25%.
With Multithreading: Number of Threads = 4

Four threads are working at 100%.
That is why the CPU Usage is around 100%.

Extensions and Units

Vocabulary for Photometry and Radiometry Units

<table>
<thead>
<tr>
<th>Language</th>
<th>Photometry</th>
<th>Radiometry</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>E (Lux)</td>
<td>I (Cd)</td>
</tr>
<tr>
<td>English</td>
<td>Illuminance</td>
<td>Intensity</td>
</tr>
<tr>
<td>French</td>
<td>Eclairément</td>
<td>Intensité</td>
</tr>
<tr>
<td>German</td>
<td>Beleuchtungsstärke</td>
<td>Lichtstärke</td>
</tr>
<tr>
<td>Italian</td>
<td>Illuminamento</td>
<td>Intensità</td>
</tr>
<tr>
<td>Japanese</td>
<td>照度</td>
<td>光度</td>
</tr>
<tr>
<td>Chinese</td>
<td>照度</td>
<td>強度</td>
</tr>
</tbody>
</table>

Extensions
A system includes different kinds of specific files (spectrum, ray file, material,...).

There are several specific files in the software:

- SPEOS input files contain all input data as surfaces, materials and spectra, created by the user or downloaded from the OPTIS online library (http://www.optis-world.com/download_software_libraries.asp) specific to the project.
<table>
<thead>
<tr>
<th>FEATURES</th>
<th>SPEOS INPUT FILES</th>
<th>EXTENSIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Optical Properties</td>
<td>Ambient Material File</td>
<td>.material</td>
</tr>
<tr>
<td></td>
<td>Simple Scattering File</td>
<td>.simplescattering</td>
</tr>
<tr>
<td></td>
<td>Surface Optical Properties Files</td>
<td>.scattering .brdf .bsdf .bsdf180 .coated .mirror .doe .fluorescent .grating .retroreflecting .anisotropic .polarizer .anisotropicbsdf .rdr .unpolished</td>
</tr>
<tr>
<td></td>
<td>Photon Map File</td>
<td>.pm</td>
</tr>
<tr>
<td>Sources</td>
<td>Spectrum File / Transmittance File</td>
<td>.spectrum</td>
</tr>
<tr>
<td></td>
<td>Intensity Files</td>
<td>.ies .ldt .intensity</td>
</tr>
<tr>
<td></td>
<td>HDRI File</td>
<td>.hdr .exr</td>
</tr>
<tr>
<td></td>
<td>Image Files</td>
<td>.bmp .jpg .png .rgb .tiff</td>
</tr>
<tr>
<td></td>
<td>Ray File</td>
<td>.ray</td>
</tr>
<tr>
<td></td>
<td>Temperature Field File</td>
<td>.OPTTemperatureField</td>
</tr>
<tr>
<td></td>
<td>Surface Optical Properties Files</td>
<td>.scattering .brdf .bsdf .bsdf180 .coated .mirror .doe .fluorescent .grating .retroreflecting .anisotropic .polarizer .anisotropicbsdf .rdr .unpolished</td>
</tr>
<tr>
<td>LCD Component</td>
<td>OppiCid File</td>
<td>.op_lbf</td>
</tr>
</tbody>
</table>
### FEATURES

<table>
<thead>
<tr>
<th>SPEOS INPUT FILES</th>
<th>EXTENSIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Image Files</td>
<td>.jpg, .png</td>
</tr>
<tr>
<td><strong>3D Texture</strong></td>
<td>3D Texture Mapping</td>
</tr>
<tr>
<td>Sensors</td>
<td>Text File</td>
</tr>
<tr>
<td></td>
<td>Distortion File</td>
</tr>
<tr>
<td><strong>Windshield Image Video Distortion</strong></td>
<td>Video File</td>
</tr>
</tbody>
</table>

- **SPEOS Output Files** are automatically created by the software after the simulation is run. They contain result files from simulations.

<table>
<thead>
<tr>
<th>SPEOS INPUT FILES</th>
<th>EXTENSIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Result Files</strong></td>
<td>XMP File</td>
</tr>
<tr>
<td></td>
<td>XM3 File</td>
</tr>
<tr>
<td></td>
<td>Light Path Finder File</td>
</tr>
<tr>
<td></td>
<td>Ray File</td>
</tr>
<tr>
<td></td>
<td>HDRI File</td>
</tr>
<tr>
<td></td>
<td>Intensity Files</td>
</tr>
<tr>
<td></td>
<td>OptisVR File</td>
</tr>
<tr>
<td><strong>Images Files</strong></td>
<td>.bmp, .jpg, .png, .rgb, .tiff</td>
</tr>
<tr>
<td><strong>Projected Grid File</strong></td>
<td>.OPTProjectedGrid</td>
</tr>
<tr>
<td><strong>Surface Optical Properties Files</strong></td>
<td>.brdf, .bsdf180, .anisotropicbsdf, .unpolished</td>
</tr>
<tr>
<td><strong>Video File</strong></td>
<td>.avi</td>
</tr>
<tr>
<td><strong>VMP Map File</strong></td>
<td>.vmp</td>
</tr>
<tr>
<td><strong>Photon Map File</strong></td>
<td>.pm</td>
</tr>
<tr>
<td><strong>Simulation Reports</strong></td>
<td>HTML File</td>
</tr>
<tr>
<td></td>
<td>CSS File</td>
</tr>
<tr>
<td></td>
<td>Image File</td>
</tr>
<tr>
<td></td>
<td>Text File</td>
</tr>
</tbody>
</table>
Changing the User Interface Language

To set the Japanese language for the user interface, you can view the CATIA V5 User’s Documentation.

The technical documentation automatically appears in English for all the selected languages.

Managing Documents

Features
A feature created by Copy/Paste inherits its name from the copied feature’s name followed by a dot and the index of the copy.

Isolated Simulation
An isolated simulation inherits its name from the original simulation followed by a dot and the index of the isolation.

Creating a Set of Features

With the Feature Set, you can group different objects of the same type in the specification tree to ease the management and search of your features. The objects are gathered into customized sub-groups, under a same node of the specification tree.

You can use the Feature Set for Simulations, Optical Properties, Sensors, Sources, Measures and Windshield Analyses.

1. Right-click an existing feature node from the specification tree.
2. Click Node Name Object.
3. Click Create Set.
   A new Set of Node Name node appears under the main feature node.
   You can right-click the created node and rename it in its properties.

Using the Other Selection Option

You must be selecting faces, volumes or solids in the selection list.

1. From the graphics view, right-click a face.
2. Select Other Selection....
   The Other Selections window opens.
3. From the list, you can go up in the face hierarchy and select a surface or a solid.

OPTIS Distributed Computing Restrictions

OPTIS Distributed Computing does not work in some cases:

- In Direct Simulation, when using intensity, irradiance, and radiance sensors with Light Expert.
  You can update, but all calculation does not run; the extension file .lfp does not appear.
- In Inverse Simulation, when using a radiance sensor with Light Expert.
  You can update, but all calculation does not run; the extension file .lfp does not appear.
- In Inverse Simulation, when using a camera sensor with Digital Vision and Surveillance.
  The extension files .xmp, .hdr, .png do not appear.
- With Virtual BSDF Bench, you cannot use OPTIS Distributed Computing.
Monitor Color Calibration

Understanding the Monitor Display Limitations

Basics of Colorimetry

What Visible Light is Made of

In physics, the color of light is defined by its wavelength, or its combination of wavelengths (spectrum). Below is white daylight spectrum:

Visible light is an electromagnetic wave with a wavelength between 380nm and 700nm. Under are UltraViolets, above are InfraReds.

How our Eyes See Colors

The human eye has three kind of cone cells to perceive color: Red (L), Green (M) and Blue (S), with particular wavelength sensibility.

Information from these cells are then combined by the brain to be perceived as color information.
How is Modeled the Human Color Vision
The CIE made a standard which allows to map any combination of wavelengths possible and the resulting color seen by the human eye:

In such model, one color is given by (x;y) coordinates though an infinite wavelength combinations can match it. A third dimension (Y) gives the brightness (dark to light).

Colorimetry Limitations
Color Limitations of a Standard Display

- Based on three colors mixing: Red Green Blue
- Displayable colors in a triangle: RGB Gamut

![RGB Gamut Diagram](image)

- Colors outside this triangle are approximated to the closest on the edge of the triangle
- Gamut changes from a display to another

**Dynamic Range Limitation (LCD Displays)**
- Standard display reaches around 200 cd/m²
- Black is around 0.5cd/m²
- Color coded on 256 levels per color
  - 16M colors, only two orders of magnitudes luminance
- Not same dynamic range for all colors
  - White reaches 200 cd/m² (R=255,G=255,B=255)
  - Red reaches around 70 cd/m² (R=255,G=0,B=0)

**Other LCD Limitations**

![CCFL display luminance map](image)

- Uneven backlight: Middle area generally brighter than sides
- CCFL warm up time: Takes 5 to 30 min to reach max luminance
- White LED color shift with time: When getting old, white LED phosphor can shift to cooler white
- Color shift at wide angle of view: Depends on LCD matrix technology

Why Calibrating a Display
RGB addressing is standard, but
- Gamut depends on display technology
  - CTR, PDP, LCD, OLED,…
  - Various LCD backlights: CCFL (flu), White LED, RGB LED
- Many LCD displays nowadays are cool white by default: 7200K instead of 6500K (neutral white).
- Display calibration will adjust the RGB addressing from the computer to match the color capabilities of the display used.

Following example are showing the difference before and after display calibration.
- Desktop display is CCFL backlight, 6000K white point.
- Laptop display is White LED backlight, 7200K white point.
Doing Color Calibration

You must buy a calibration probe to calibrate your display.

1. Install the probe software on your computer.
2. Restart your computer, plug the probe to USB and wait for driver recognition.
3. Launch the probe software.
4. Place the probe software window in the display to calibrate.
5. Follow the calibration instructions.

Doing the Caracterization

After the calibration, an ICC profile of the display is created.

The ICC profile saves the display capabilities of the screen. It is used to adjust the RGB data sent by software to the screen so the color displayed matches the intent.

One ICC profile correspond to a single combination of display, display settings, graphic adapter and computer.

1. Select the ICC profile.
   Windows 7 can manage several ICC profiles from Control Panel\Color Management.
2. Set as default the profile you wish to use in the list for the selected display.
   The profile is loaded at Windows startup.
3. Apply the ICC profile.
   The software can include a tool to force ICC profile loading.
4. Loads and apply the default profile for all display devices.

Matching Color between XMP Results and Calibrated Display

A calibration verification is required.

1. Display the xmp result on the monitor to measure.
2. With a photometric camera, take pictures of the monitor and check the colorimetric coordinates for each color.
   Photometric camera has to be perfectly aligned with the screen normal.
   It is recommended to measure color saturated areas to remove any dependencies with the ambient light of the room where is measured the screen.
3. Compare the x and y parameters of the xmp colorimetric data (xyY) to the colorimetric data obtained with the photometric camera.
4. Compare for display in Virtual Photometric Lab.
FEATURES

Optical Properties

You must have built a material thanks to editors before applying features to it.

Optical properties define how light rays interact with geometries.
A material should be applied on a geometrical element or created in a material library.
Materials can be enhanced with Volume Optical Properties (VOP) and Surface Optical Properties (SOP).

- **Volume Optical Properties (VOP)** define the behavior of light rays when they are propagated in a solid.
  - Volume Optical Properties can be defined on a material created with the Material Editor (Editors).
  - With this model you can build complex materials and store them in a library.
- **Surface Optical Properties (SOP)** define the behavior of light rays when they hit a face.
  - Surface Optical Properties can be defined on a material created by the Surface Optical Properties Editor.
  
  Note that Simple Scattering Surface Editor (Editors) and Advanced Scattering Surface Editor (Editors) are the two main optical properties editors.

Note that you can download materials from the OPTIS Online Library (http://www.optis-world.com/download_software_libraries.asp) by clicking Online Library (Help).

Face Optical Properties

Creating Face Optical Properties

When a material is applied to a geometry, all the faces of this geometry have the same Surface Optical Properties (SOP).
With Face Optical Properties, you can apply different Surface Optical Properties to a set of faces.

1. Click Face Optical Properties (Optical Properties).
2. Click the faces in the graphics area.
   - Note that selected faces must have other Surface Optical Properties than the applied material.
3. Set the parameters (see page 19).
   - You can click Preview to preview the surface optical property in the 3D view.
4. Click OK.
   - The surface optical property appears in the specification tree.
   - The surface optical property appears in the 3D view.
Parameters of Face Optical Properties

- In the Surface Optical Properties (SOP) group box, you must select the type of Surface Optical Properties you want to apply to the face.
  - If you select Mirror, in the Reflectance box, you can type or edit the reflectance value.
  - If you select Library, you must browse a surface optical properties file.

You also can edit the file by clicking Edit....

When selecting a BSDF180 file, the normal direction corresponding to the Normal BSDF can be oriented selecting the surface and using the Reverse direction button. The BSDF 180 supports the anisotropy. For more details, you can view BSDF180 Surface.

- You can select the Mouse-over multi-selection mode check box to easily selecting a lot of faces.

You must click the first element to start the selection, move the cursor over every needed face, and then click the last element when the selection is completed.

If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

Editing the Optical Properties of a Material

You must select a geometry and click Edit SPEOS properties (Tools) to access the optical properties of a material.

Volume Optical Properties (VOP)

- You can select Opaque for non transparent material.
- You can select Optic for transparent colorless material without bulk scattering.
  - In the Index box, you can type or edit the index value.
  - In the Absorption box, you can type or edit the absorption value.
  - From the Constringence box, you must select true or false.
    - If you select true, you can type or edit the contingencies of value.


- You can select Library to browse a .material file.
  You also can select the file and click Edit... to edit it.

For more details about the .material file, you can view Using the User Material Editor.

Surface Optical Properties (SOP)

- You can select Mirror for perfect specular surface of non transparent material.
  In the Reflectance box, you can type or edit the reflectance value.
- You can select Optical polished for perfect polished surface of transparent material.
- If you select Library, you must browse a surface optical properties file.
  You also can select the file and click Edit... to edit it.

Note that material data could be stored either in a CATPart/CATProduct file or a CATMaterial file according to the Link to file check box status when the material is applied.

If the Link to file option is used when the material is applied, the external links will be relative to the CATMaterial file.

If you want to use materials without optical properties, you can click Remove.
To reset optical properties, you can click Reset Optical Properties.

To edit the optical properties of a material, you can also, from a material in the CATIA specification tree, right-click, select Properties, and then click More... to edit the Optical Properties tab.
**Ambient Material**

**Creating an Ambient Material**

Ambient Material enables the simulation of light behavior in media such as water or fog. The use of ambient material in inverse simulation is not compatible with the use of ambient sources.

1. Click Ambient Material (Optical properties).
2. Set the parameters (see page 20).
3. Click OK.

The ambient material appears in the specification tree. The ambient material appears in the 3D view.

![Simulation without ambient material](image1) ![Simulation with ambient material](image2)

**Parameters of an Ambient Material**

From the Type list in the Volume optical properties (VOP) group box, you must select the type.

- You can select Optic for transparent colorless material without bulk scattering.
  - In the Index box, you can type or edit the index value.
  - In the Absorption box, you can type or edit the absorption value.
  - From the Constringence box, you must select true or false.
    - If you select true, you can type or edit the constringence value.
  

- You can select Library to browse a .material file.
  - You also can edit the file by clicking Edit....

  For more details about the .material file, you can view Using the User Material Editor.

**3D Texture**

**Overview**

Some specific applications as Light Guide or BEF films are composed of a millions geometrical items (Patterns). These ones are duplicated according to a specific distribution (Mapping) on a base geometrical item (Support). The duplicated texture can be added, removed... (Boolean Operation), can have optical properties (Volume or Surface) identical to or different from the Support.
As the OPTIS software is not able to model them geometrically, the 3D texture tool interest is to model them for an optical simulation without having to create them geometrically in the CAD model. 3D Texture is thus dedicated only to the modeling of the geometry of a huge number of similar elements having the same shape, with no intersection of patterns between them.

Main Capabilities
- Patterns can be applied on any CAD shapes.
- 3D Texture can be applied on any CAD surface (flat or free form, rectangular or not).
  - You cannot apply a 3D texture on an element having a surface tangential to another element.
- Boolean operations are Add on same material, Add on different material, Add in, Insert, Remove.
- The number of patterns is up to hundreds of millions.
- The low memory usage is around 150 Mb for 1 million patterns.
- 3D Texture can be optimized thanks to VBScript.
- Optimization using optimization tool or VBScript.
- The surface optical property quality of the pattern can be different from the support surfaces.
- The material quality of the pattern can be different from the support surfaces (ink jet).
- The position and the orientation can be different for each pattern (constant or variable mapping).
- The Pattern size can be different for each pattern.

Creating a 3D Texture

A pattern for 3D texture and a mapping file must be created.

It is strongly recommended to check the behavior of the 3D textures used in an Interactive Simulation.

1. Click 3D Texture (Optical Properties).
2. Set the parameters (see page 21).
3. Click OK.

A tutorial to create a 3D Texture is available.

Parameters of 3D Texture

Axis System Definition

An axis system is defined by one point and two lines. Position, orientation, scaling of each pattern will be defined for this axis system for the mapping file. For other mappings the xy plane is used as a reference for the offset. The z direction is used to be the direction for the projection of the mapping filtering.

To define the axis system, you must click one point and two lines in the graphics area.

Support

The Support is a Part on which all patterns for all types of mapping files are projected. A boolean operation is executed between Part support and patterns. In Support box, you must select a geometry support on which the 3D texture is applied.

Materials are added or removed from this geometry.

The support can be flat or freeform.

Pattern

The pattern is a geometrical item that can be duplicated. A CATIA part can be inserted as a Pattern.

To be selected as a pattern a part has to be included in the working assembly.

- In Part File box, you must browse to select a CATIA V5 part file.
To be selected as a pattern a part must be included in the working assembly.

- In Scale box, you can set the global scale of the Pattern. It is a 1 Scale Factor.
  This global scale factor is cumulative to each pattern scale factor (see page 30) (Final pattern scale factor =
  global scale factor * pattern scale factor).
  The value must be different from 0.

Then, the .OPT3DMapping file is:

- x1, y1, z1, i1x, i1y, i1z, j1x, j1y, j1z, k1
- x2, y2, z2, i2x, i2y, i2z, j2x, j2y, j2z, k2
- x3, y3, z3, i3x, i3y, i3z, j3x, j3y, j3z, k3
  ...

You can select the Preview meshing check box to visualize the patterns contained in a parallelepiped or a sphere.

The 3D texture preview can be tuned using the zone selection tool's arrows, edges and central point.
By clicking on arrow and moving the mouse along its direction, you can resize the zone selection tool along X, Y or
Z direction.
By clicking on the central point and one edge of the zone selection tool and moving the mouse, you can switch the
zone selection tool in the 3D view.

**Operation**

In Operation list, you must select a boolean operation.

You can view Setting the Simulation Properties (see page 138).
The following table describes the different Boolean Operation types.
For all the operations, you cannot create intersections between patterns.

Add On
You cannot set tangent surfaces between patterns and a support.

When using the Add on boolean operation with a tangent surface, a gap is needed between the texture and this tangent surface. The gap must be bigger than ten times the Geometrical Distance Tolerance value.

For more details, you can view Simulation Properties (see page 138).
Add In
You cannot set tangent surfaces between patterns and a support.

Remove
You cannot set tangent surfaces between patterns and a support.

When using G for the small gap, you must set the Geometrical Distance Tolerance to G / 100 in the assembly preferences (ex: if G=1e-5 then Geometrical Distance Tolerance=1e-7). This gives fewer errors in the propagation of the photons.

The texture width cannot be larger than the material width. Here, you must create two add on textures, one for each material side.

All available operations take into account the Surface Optical Properties of the pattern.
Add on different material, Insert and Add in use the Volume Optical Properties of the pattern.
Add on same material uses the Volume Optical Properties of the support geometry.
Remove uses air.

The diffuse supports and patterns are compatible with the operations as follows:

<table>
<thead>
<tr>
<th></th>
<th>Diffuse Support</th>
<th>Diffuse Pattern</th>
</tr>
</thead>
<tbody>
<tr>
<td>Remove</td>
<td></td>
<td>Not applicable</td>
</tr>
<tr>
<td>Add on Same Material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add on Different Material</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Add In</td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Insert</td>
<td>✓</td>
<td></td>
</tr>
</tbody>
</table>

Mapping
In the Mapping box, you can:
- Select From file if you want to generate 3D texture distributions using an existing file.
- Select From definition if you want to automatically generate 3D texture distributions.

Mapping File
1. In the Mapping box, you can select From file to define the mapping from a file.
2. In File, browse to select an .OPT3DMapping file.

Pattern Numbers: x y z ix iy iz jx jy jz kx ky kz
The first line corresponds to the number of patterns that you are going to represent in the texture.
x y z: Coordinate of the pattern's origin in the texture coordinate system.
ix iy iz: Orientation of the pattern with respect of the X direction of the texture coordinate system.

jx jy jz: Orientation of the pattern with respect of the Y direction of the texture coordinate system.

kx, ky, kz: Pattern scale values for respectively x, y, z directions. "1" value means 100% of the original pattern size.

**Automatic Mappings**

To easily and automatically generate 3D texture distributions, without a mapping file, you can select From definition.

You can select one of the four types of mapping available:

- Rectangular mapping (see page 25).
- Circular mapping (see page 26).
- Hexagonal mapping (see page 27).
- Variable pitches mapping (see page 28).

Each type describes a way to create a virtual grid that is going to be projected on a part’s surface.

1. A virtual grid is created using standard parameters (distance between patterns, mapping length...).
2. A Quilt or a Face is used to define the grid limitations.
3. All the patterns included in the limited grid are projected along the Z direction on the first encountered surface of the selected part.
4. A shift along Z could be applied on the projected patterns.

**Rectangular Mapping**
The following table describes the mapping parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Distance between patterns along X Direction</td>
<td>Element limiting the area (see page 29) on which the mapping is going to be created (optional)</td>
</tr>
<tr>
<td>Mapping length along X Direction</td>
<td>You can select the following elements. Directly click them in the graphics area, or use the Other Selection option (see page 11).</td>
</tr>
<tr>
<td>Angle with X Direction</td>
<td></td>
</tr>
<tr>
<td>Mapping length along Y Direction</td>
<td></td>
</tr>
<tr>
<td>Distance between patterns along Y Direction</td>
<td></td>
</tr>
<tr>
<td>Angle with Y Direction</td>
<td></td>
</tr>
</tbody>
</table>

**Filter Surface**

- **Faces**
- **Surfaces**
- **Solids**

**Shift Surface**

- Shift surface along Z Direction allowing an offset (see page 30) (optional)

**Pattern Direction**

- Position of the pattern (see page 30) according to different parameters

**Pattern X, Y, Z Scale**

- Pattern scale (see page 30) along X, Y and Z directions

*Circular Mapping*
The following table describes the mapping parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial distance between patterns</td>
<td>Element limiting the area (see page 29) on which the mapping is going to be created (optional)</td>
</tr>
<tr>
<td>Mapping radius</td>
<td>You can select the following elements. Directly click them in the graphics area, or use the Other Selection option (see page 11).</td>
</tr>
<tr>
<td>Distance between two rings</td>
<td></td>
</tr>
<tr>
<td>Angle with X Direction</td>
<td></td>
</tr>
<tr>
<td>Pattern Direction</td>
<td>Position of the pattern (see page 30) according to different parameters</td>
</tr>
<tr>
<td>Pattern X, Y, Z Scale</td>
<td>Pattern scale (see page 30) along X, Y and Z directions</td>
</tr>
</tbody>
</table>

**Filter Surface**

- **Faces**: ✔️
- **Surfaces**: ✔️
- **Solids**: ✗

**Shift Surface**: Shift surface along Z Direction allowing an offset (see page 30) (optional)

**Hexagonal Mapping**
### Mapping length along X Direction

### Angle with X Direction

### Mapping length along Y Direction

### Angle with Y Direction

### Hexagon length along X Direction

### Hexagon length along Y Direction

### Hexagon edge length along X Direction

### Distance between hexagons along X Direction

**Central Point** Display or hide the central point of hexagons

**Regular Mapping** Activate the regular distance between hexagons along X Direction

Element limiting the area (see page 29) on which the mapping is going to be created (optional)

You can select the following elements.

Directly click them in the graphics area, or use the Other Selection option (see page 11).

<table>
<thead>
<tr>
<th>Filter Surface</th>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✗</td>
</tr>
</tbody>
</table>

**Shift surface** Shift surface along Z Direction allowing an offset (see page 30) (optional)

**Pattern Direction** Position of the pattern (see page 30) according to different parameters

**Pattern X, Y, Z Scale** Pattern scale (see page 30) along X, Y and Z directions

---

**Variable Pitches Mapping**

**Mapping View**

The following table describes the mapping parameters.

<table>
<thead>
<tr>
<th>Mapping length along X Direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle with X Direction</td>
</tr>
</tbody>
</table>
Mapping length along Y Direction

Angle with Y Direction

X pitch curve

Scale of X pitch curve

Y pitch curve

Scale of Y pitch curve

Filter Surface

Element limiting the area (see page 29) on which the mapping is going to be created (optional).
You can select the following elements. Directly click them in the graphics area, or use the Other Selection option (see page 11).

<table>
<thead>
<tr>
<th>Filter Surface</th>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shift surface</td>
<td>✔️</td>
<td>✔️</td>
<td>✗</td>
</tr>
<tr>
<td>Pattern Direction</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shift Scale</td>
<td>Pattern scale (see page 30) along X, Y and Z directions</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The X pitch curve must cut the yOz plane of the 3D texture 3D, and Y pitch curve must cut the xOz plane of the 3D texture.

Mapping Filtering

The following table describes the Surface Filtering View.

3D TEXTURE MAPPING
FILTERING WITH FLAT SUPPORT

SHAPE DEFINITION WITH SPEOS CAA V5 BASED
**Surface Shift**

With the shift surface, you can add an offset along the Z direction to the projected patterns.

You can set a value to define the ratio applied to the offset calculated for each pattern.

**Pattern Direction**

You have two different ways to position the pattern.

- If you want to orientate the pattern according to the support of the 3D Texture, select **Constant**.

- If you want to orientate the pattern according to the normal of the surface, select **Normal to Support**.

**Pattern Scale**

You can define three scale factors to set the size of each pattern independently on the 3 axes.

For each direction, you can set in the Pattern X Scale, Pattern Y Scale and Pattern Z Scale group boxes:
- A surface. The scaling factor to apply to a specific mapping point is defined by the height of the point of this surface, at the corresponding coordinates (X,Y).
- A scale value, applied uniformly on the direction.

3D Texture scaling along the X, Z and Y axes

The pattern scale factors are cumulative to the global scale factor (see page 21) (Final pattern scale factor = global scale factor * pattern scale factor).

**LCD Component**

**Creating a LCD Component**

*With the LCD component, you can use OptoPartner’s LuCiD LCD stack in SPEOS CAA V5 Based. LuCiD is required and must be installed.*

LCD is limited to SPEOS CAA V5 Based 32 bits for 32 bits Operating System and SPEOS CAA V5 Based 32 bits for 64 bits Operating System.

A LCD component is both an optical property and a geometry. It has to be selected as a geometry for a simulation.

1. Click LCD component (Optical Properties).
2. Set the parameters (see page 32).
You can click Preview to preview the component in the 3D view.

3. Click OK.
   The LCD Component appears in the specification tree.
   The LCD Component appears in the 3D view.

![LCD component](image)

Parameters of a LDC Component

- In the Axis system group box, you must set the parameters.
  - The axis system sets the position and the orientation of the LCD component in the space.
  - To select one point in the Origin box, and two lines in the X Direction and Y Direction boxes, you must click them in the graphics area.
  - You can click Reverse direction to reverse the direction.
- In the Image box, you must browse a .jpg or .png file.
  - You can also edit the file by clicking Edit…
  - The image file corresponds to the image displayed on the LCD.
- In the LCD box, you must browse an .op_lbf file.
  - You can also edit the file by clicking Edit…
  - The LCD file specific to LuCiD contains the definition of the optical properties of the LCD and the surface geometry.

Polarization Plate Component

Polarization plate is used to define polarization components like polarizers, wave plates, or any component whose Jones matrix is known.

Radiance sensor with direct simulation and gathering with inverse simulation and fast transmission caustics are not compatible with polarization plates or polarizers.

Ideal Plates

To define an ideal plate, you can select a standard from the list of proposed components, or give the Jones matrix of your component in a text file.

The plate has the same effect for any light ray because the Jones matrix is constant.

Plates with Wavelength and Incident Angle Dependency

Every polarization component, except depolarizing components, can be divided into a diattenuator and a retarder.

For example, a linear polarizer is a diattenuator only and a quarter wave plate is a retarder only.

This decomposition is used to take wavelength and angle dependency into account.

In SPEOS CAA V5 Based, the wavelength and angle dependency are only carried by the retarder because the retarder works on a principle of phase shift.
For plates with wavelength and angle dependency, you must define a retarder. This retarder requires accurate data on its optimal wavelength, the plate order and the birefringent material from which it is made.

Defining a diattenuator is optional but recommended.

In case you need to use the polarization plate with wavelength and incidence dependency in an inverse simulation, check in the simulation properties that:

- It is a Monte-Carlo simulation.
- Dispersion is set to True.

Creating a Polarization Plate Component

1. Click Polarization Plate (Optical Properties).
2. Set the parameters (see page 33).
   - You can click Preview to preview the component in the 3D view.
3. Click OK.
   - The Polarization Plate appears in the specification tree and in the 3D view.

Parameters of a Polarization Plate Component

Axis System

In the Axis system group box, you must set the parameters.

The axis system sets the position and the orientation of the polarization plate in space.

To select one point in the Origin box, and two lines in the X Direction and Y Direction boxes, you must click them in the graphics area.

You can click Reverse direction to reverse the direction.

In many cases, the Jones matrix is different when the plate is illuminated on one side or on the other. To identify which side corresponds to the Jones matrix you are defining, take the local base of the plate. Your matrix will be used for the plate illumination in the direction of the Z axis of this base, making an angle smaller than 90° with it. If light arrives from the other side (around -Z), the Jones matrix for the reversed component will be automatically computed and used instead. If your Z axis is not in the general direction in which light is expected to travel, use the "Reverse direction" option for X or Y in order to modify Z. Note that for a retarder alone, the matrix remains the same when the plate is turned.

Plate Geometry

In the X and Y group boxes, you can type or edit coordinates of the Start and End points of the plate rectangle.

You can select the Mirrored Extent check box to link the Start and End values and have light from all space.

Wavelength and Incidence Dependency

Before going further, you must decide if the plate has a dependency on wavelength and incident angle, that is if the plate is a polarizer or a retarder with a diattenuator.

Select the Wavelength & incidence dependency check box according to the dependency you need.

- If you do not select Wavelength and incidence dependency, the Polarization field appears.
- If you select Wavelength and incidence dependency, the Retarder and Diattenuator fields appear.

Polarization

In the Type group box, select a polarizer type:

- You can select Library.
  - Select a file from the library or choose your own .polarizer file.
  - You can also click Edit to open and edit the file in the Polarizer Surface Editor.
  
  The axes specified in the file (lines 3 and 4) are not used. The axis selected with the mouse is used instead.
  
  For a rotated component, apply the rotation to the Jones matrix.

- You can select Linear polarizer (with principal axis along X of the local base).

- You can select Left circular polarizer.

- You can select Right circular polarizer.
You can select Half wave plate (with fast axis along X of the local base).

You can select Quarter wave plate (with fast axis along X of the local base).

If you select Linear Polarizer, Half wave plate or Quarter wave plate, you can set the orientation angle of the polarizer in the Orientation spinbox.

**Diattenuator**

Defining the diattenuator is optional. If you define a diattenuator, it is placed before the retarder, considering the local Z direction.

A diattenuator is a component whose transmittance depends on the incident polarization.

In the Diattenuator group box, select a diattenuator type:

- You can select Library.
  - Select a file from the library or choose your own .polarizer file.
  - You also can edit the file by clicking Edit…
  - The axes specified in the file (lines 3 and 4) are not used. The axis selected with the mouse is used instead.
  - For a rotated component, apply the rotation to the Jones matrix.

- You can select Left circular polarizer.
- You can select Right circular polarizer.
- You can select Linear Polarizer.

If you select Linear Polarizer, you can set the orientation angle of the polarizer in the Orientation spinbox.

**Retarder**

Defining the retarder is mandatory.

A retarder is a piece of birefringent material dephasing (i.e. retarding) the two orthogonal parts of the electric field one compared to the other.

In the Retarder group box,

1. In the File group box, click Select and browse a .material file.
   - You also can edit the file by clicking Edit…
   - The birefringent material is given by the .material file. It has to be an unaxial material. Two of the three refractive indices na, nb, nc must be equal. For a positive material, na = nb < nc. For a negative material, na < nb = nc.
   - If the three indices are not in increasing order, you will get an error.
   - Vectors I and J are not used. The X direction you choose for the plate is the direction of na. The Y direction is the direction of the first index different of na, so that the extraordinary axis is always in the plane of the input face.

2. In the Optimal Wavelength (w) box, type or edit the wavelength value.
   - The optimal wavelength is the wavelength for which the retarder plate has been designed.

3. In the Retardance group box, you must give the N and T values in the corresponding fields, knowing that the total Retardance = N * w + w / T.
   - N must be an integer and T a real number.
   - The plate induces the retardance (in waves) given by w/T. For a quarter wave plate, T=4. If the plate is a multi-order wave plate, the order N is defined so that the total retardance is N*w+w/T.
   - The plate thickness in mm is computed using the retardance and the refractive indices, and appears in the Thickness group box.

---

**Using Rendering Properties as Optical Properties**

To use Rendering Properties as Optical Properties, you must launch an inverse simulation. From the inverse simulation in the specification tree, you must right-click, select Properties, click More…, select
the Inverse Simulation tab, and then select true from the Authorize the use of rendering properties as optical properties list. From a material in the CATIA specification tree or a face optical properties in the specification tree, you must right-click, select Properties, click More… to edit the Optical Properties tab, and then you must click Remove.

Physical parameters are automatically converted from the Rendering tab of the material according to the following conversion table.

<table>
<thead>
<tr>
<th>RENDERING PARAMETERS RP</th>
<th>PHYSICAL PARAMETERS PP(α)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ambient [0, 1] + RGB</td>
<td>½ Lambertian L(α)</td>
</tr>
<tr>
<td>Diffuse [0, 1] + RGB</td>
<td>½ Lambertian L(α)</td>
</tr>
<tr>
<td>Specular [0, 1] + RGB</td>
<td>Gaussian G(α)</td>
</tr>
<tr>
<td>Emission [0, 1] + RGB</td>
<td>None</td>
</tr>
<tr>
<td>Roughness [0, 1]</td>
<td>Gaussian Angle α</td>
</tr>
<tr>
<td>Transparency [0, 1] + RGB</td>
<td>Transmitted Specular ST(α)</td>
</tr>
<tr>
<td>Refraction</td>
<td>Refractive Index n</td>
</tr>
<tr>
<td>Reflectivity [0, 1] + RGB</td>
<td>Reflected Specular SR(α)</td>
</tr>
</tbody>
</table>

Optical properties can be added to the CATIA V5 default library but be aware that optical properties will be lost at the next CATIA V5 update.

Sources

Surface Source

Creating a Surface Source

With a surface source, you can model the light emission of a source taking into account its physical properties as the flux, the spectrum, the emittance and the intensity. A surface source can be defined with any face in a geometry which emits rays.

1. Click Surface Source (Sources).
2. Set the parameters (see page 36).
   - You can click Preview to preview the source in the 3D view.
3. Click OK.
   - The surface source appears in the specification tree.
   - The surface source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.
   - You can edit the surface source properties (see page 58).
Parameters of a Surface Source

Surface source with ray files emission cannot be used in inverse simulations.

Flux
In the Flux group box, you can choose the unit and the value of the flux.
If you set Library as Intensity type, a From file check box appears. Select it to assign automatically the value from the library file.

The power of the source can be defined in Watt, a radiometric unit, or in Lumen, a photometric unit.

Spectrum
In the Spectrum group box, you can select the type of spectrum.
You can select a monochromatic source, a blackbody or a .spectrum file from the library.

The .spectrum file can be a material created with the Spectrum Editor (Editors).
• If you select Monochromatic, you can type or edit the wavelength value.
• If you select Blackbody, you can type or edit the temperature value.
• If you select Library, you must browse a .spectrum file.
  You can also edit the file by clicking Edit…

If you select a XMP map with spectral conoscopic intensity in the Library group box, the spectral information of the map is displayed in the Spectrum group box.

Emittance
In the Emittance group box, you can select the different following elements:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11).
You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.
If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

Intensity
In the Intensity group box, you can select the intensity diagram.

The intensity diagram of a source describes in which directions is made the emission.

Lambertian
The simplest model is Lambertian which is a distribution law given by cos (theta).

The default value 180 degrees is for one complete hemisphere.
The intensity formula for Lambertian is \( I = \cos(\theta) \)
Cos: \( I = \cos(\theta) \).
The following image shows a luminance map of a source with the Lambertian law for an intensity distribution.

A lambertian surface source with angle equal to zero has parallel rays.

**Cos**

If you click Cos,
- In the Total angle box, you can type or edit the value,
- In the N box, you can type or edit the value.

The following image shows a luminance map of a source with the Cosnθ law for an intensity distribution.

**Gaussian**

The intensity formula for Gaussian is \( I = \exp\left(-\left(\frac{\theta}{a}\right)^2\right) \).

\( a \) is calculated in a way that the FWHM (Full Width at Half Maximum) angle of the Gaussian is the one given by the user.

For more details about the FWHM, you can view Parameters of Lab/Gloss Surface Properties.

When the Gaussian is not symmetric, there is a different \( a \) on both axes.

Theta is always the angle to the surface's normal.

If you click Gaussian, you can select Symmetric gaussian or Asymmetric gaussian.
- If you select Symmetric gaussian type, you can define the total angle and the FWHM angle values by typing or editing them.
- If you select Asymmetric gaussian, you can:
  - Define the total angle by typing or editing it.
  - Define the FWHM angle X the and the FWHM angle Y by typing or editing them.
  - Choose the X and Y directions by clicking in the graphics areas.

For asymmetric gaussian type, if both FWHM X and Y angle values are equal, the axis selection is optional.

The axis can be global or local:
- Global axis: Orientation of the intensity diagram is related to the axis system.
- Local axis: Orientation of the intensity diagram is related to the normal at the surface.
If no axis is selected, the axis is local.

The following image shows a luminance map of a source with the gaussian law for an intensity distribution.

Total angle for Gaussian type is an additional parameter to shape the emission diagram. Changing this parameter changes the intensity of the source.

<table>
<thead>
<tr>
<th>FWHM = 15DEG</th>
<th>FWHM = 45DEG</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOTAL ANGLE</td>
<td>TOTAL ANGLE</td>
</tr>
<tr>
<td>=20DE</td>
<td>=75DE</td>
</tr>
</tbody>
</table>

To keep a constant intensity on the axis, one would have to tune the source power manually.

**Library**

The distribution can also be given by an intensity distribution file in an IESNA, an Eulumdat, an Extended Map or an OPTIS Intensity format.

If you click Library, in the File box, you can browse a .ies, .ldt, .xmp or .intensity file. You can also edit the file by clicking Edit...

You must select the X and Y directions to set up the axis system. The axis can be global or local:

- Global axis: Orientation of the intensity diagram is related to the axis system.
- Local axis: Orientation of the intensity diagram is related to the normal at the surface.

**Associated Geometries**

The emissive faces of a light source are associated to a lot of geometrical bodies which play a role in the optical system.

- By clicking More, you can select the different following elements to take them into account in the simulation:
You must have applied V5 materials and set the optical properties.
For N light sources with X bodies, you must select N x X bodies.

```
Faces  Surfaces  Solids

Directly click the elements in the graphics area, or use the Other Selection option (see page 11).
```

- Light source definition permits to take into account elements influencing intensity distribution as for example a bulb in the case of a light bulb or a lens in the case of a LED.
  If you selected the Library intensity type, you can select geometries influencing intensity distribution in the Exit Geometries list.
  Those geometries are present during source measurement and could influence its near and far field optical behavior.

```
Intensity distribution without specific exit geometry

Intensity distribution with lens defined as exit geometry.
```

You can select the following elements:

```
Faces  Surfaces  Solids

Directly click the elements in the graphics area, or use the Other Selection option (see page 11).
```

You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.

If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.
**Ambient Sources**

With an Ambient Source, you can have access to the model of environment light such as sky and sun.

You can use an ambient source with an inverse simulation.
You can view Setting the Inverse Simulation Properties. (see page 126)

When ambient sources are enabled for the direct simulation, only 2D and 3D illuminance/irradiance sensors are taken into account.

**Ambient Source with Uniform Type**

**Creating an Ambient Source with Uniform Type**

1. Click Ambient Source (Sources).
2. Select Uniform from the Type list.
3. Set the parameters (see page 40).
   - You can click Preview to preview the source in the 3D view.
4. Click OK.
   - The ambient source appears in the specification tree.
   - The ambient source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

**Parameters of an Ambient Source with Uniform Type**

**Zenith**

To set the zenith orientation, you must click a line in the graphics area.
You can click Reverse direction to reverse the direction.

**Properties**

- In the Luminance box, you can type or edit a value.
  - The value usually varies in the 1000 - 20000 cd/m² range.
- In the Spectrum box, you must browse a .spectrum file.
  - You can also edit the file by clicking Edit ....
    - Basically, the spectrum can be defined as a blackbody at 25000.0 Kelvins.
    - This can be created with the spectrum generator tool of the Spectrum Editor.
- You can activate the Mirrored extent to get an ambient light from all the space.
  - You can also deactivate it to get an ambient light only in the upper half space.
- You must activate the Sun check box to add automatically the sun to the ambient source. Then, you must click a line in the graphics area to set a sun orientation. You can click Reverse direction to reverse the direction.

The sun arrow points the sun so the arrow is in the opposite of the photons light from the sun.

The sun of the Uniform Ambient Source changes of power and color belong to its orientation.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Appeared diameter</td>
<td>0.0087 radians</td>
</tr>
<tr>
<td>Illuminance at the ground</td>
<td>Around 105 Lux (regardless its orientation)</td>
</tr>
<tr>
<td>Color temperature</td>
<td>Approximately 5800 K</td>
</tr>
<tr>
<td>Apparent diameter</td>
<td>Around 0.5 degrees</td>
</tr>
<tr>
<td>Solid angle within the one we see</td>
<td>$2 \pi (1 - \cos(0.5^\circ/2)) = 5.98 \times 10^{-5}$ steradian</td>
</tr>
<tr>
<td>Luminance</td>
<td>Around $105/5.98 \times 10^{-5} = 1.6 \times 10^9$ cd/m²</td>
</tr>
</tbody>
</table>

A sun luminance and the spectrum of an ambient source with a uniform type is equivalent to the sun luminance and the spectrum of an ambient source with a Natural Light type with a turbidity of 1, when the observer is located in London at 12am, the 30th of May.

If you want to use sun only in simulations, you must set the luminance of the uniform source to zero Candela per square meters.

**Ambient Source with CIE Standard Overcast Sky Type**

**Creating an Ambient Source with CIE Standard Overcast Sky Type**


1. Click Ambient Source (Sources).
2. Select CIE Standard Overcast Sky from the Type list.
3. Set the parameters (see page 42).
   - You can click Preview to preview the source in the 3D view.
4. Click OK.
   - The ambient source appears in the specification tree.
   - The ambient source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

*Ambient source with CIE standard overcast sky type*
Parameters of an Ambient Source with CIE Standard Overcast Sky Type

Zenith
To set the zenith orientation, you must click a line in the graphics area.
You can click Reverse direction to reverse the direction.

Properties
- In the Luminance box, you can type or edit the luminance value.
  This value usually varies in the 1000 - 20000 cd/m² range.
- In the Spectrum box, you must browse a .spectrum file.
  You also can edit the file by clicking Edit ....
  Basically, the spectrum can be defined as a blackbody at 25000.0 Kelvins.
This can be created with the spectrum generator tool of the Spectrum Editor.

Ambient Source with CIE Standard General Sky Type

Creating a Ambient Source with CIE Standard General Sky Type
This sky model is based on the publication of the CIE:

1. Click Ambient Source box (Sources).
2. Select CIE Standard General Sky from the Type list.
3. Set the parameters (see page 42).
4. Click OK.
   The ambient source appears in the specification tree.
   The ambient source appears in the 3D view.
   You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

In the Luminance box, you can type or edit a luminance value.

In the Time zone and location group box, you can select a location from the list. Note that a location can be defined manually by selecting the last line as location called User. In the Earth coordinates group box, you must type or edit values.

In the Date and time group box, you can type or edit values.

Ambient Source with Natural Light Type

Creating an Ambient Source with Natural Light Type

1. Click Ambient Source (Sources).
2. Select Natural Light from the Type list.
3. Set the parameters (see page 43).
   You can click Preview to preview the source in the 3D view.
4. Click OK.
   The ambient source appears in the specification tree.
   The ambient source appears in the 3D view.
   You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

Parameters of an Ambient Source with Natural Light Type

Zenith
To set the zenith orientation, you must click a line in the graphics area.
You can click Reverse direction to reverse the direction.

Properties

- In the Turbidity box, you must type or edit a value.
  This value usually varies in the 1 - 15 range, even if it can be greater.
  The smaller the turbidity is the clearer the sky is.
  For more details about turbidity, you can view Using Turbidity for an Ambient Source (see page 149).

- From the Sun type list, you must select Automatic or Direction.
  - If you click Automatic, you must click a line in the graphics area to set the North direction.
You can click Reverse direction to reverse the direction.

The sun direction is automatically calculated thanks to the north direction, the location on the earth, the date and the time.

- If you click Direction, you must click in the graphics area to set the sun direction.
You can click Reverse direction to reverse the direction.

Night sky model is available taking into account: Moonlight, starlight, zodiacal light and airglow.

References:
- Absolute photometry of the zodiacal light (A.C Levasseur-Rourd, R. Dumont). Aeronomie Service of the CNRS.

The sun arrow points the sun so the arrow is in the opposite of the photons light from the sun.

Sun properties:

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Appeared diameter</td>
<td>0.0087 radians</td>
</tr>
<tr>
<td>Illuminance at the ground</td>
<td>Around 105 Lux (regardless its orientation)</td>
</tr>
<tr>
<td>Color temperature</td>
<td>Approximately 5800 K</td>
</tr>
<tr>
<td>Apparent diameter</td>
<td>Around 0.5 degrees</td>
</tr>
<tr>
<td>Solid angle within the one we see</td>
<td>$2\pi(1-\cos(0.5^\circ/2))=5.98\times10^{-5}$ steradian</td>
</tr>
<tr>
<td>Luminance</td>
<td>$105/5.98\times10^{-5}=1.6\times10^{9}$ cd/m²</td>
</tr>
</tbody>
</table>

A sun luminance and the spectrum of an ambient source with a uniform type is equivalent to the sun luminance and the spectrum of an ambient source with a natural light type with a turbidity of 1, when the observer is located in London at 12am, the 30th of May.

The spectral range for the ambient source is from 380 to 780 nm for day, and from 380 to 1000 nm for night.

If you want to use sun only in simulations, you must set the Sky parameter to false. For more details, you can view Managing the Sub-Trees. (see page 57)

**Ambient Source with Environment Type**

**Environment Type Overview**

**Conventions**
- For all the different north selected types, Zenith defines the main direction for the ambient source.
- If the North is not perpendicular to the Zenith, it is projected in the perpendicular plan.
Longitude/Latitude Map Type

Light Probe Type

Horizontal Cross Type
Creating an Ambient Source with Environment Type

1. Click Ambient Source (Sources).
2. Select Environment from the Type list.
3. Set the parameters (see page 47).
   - You can click Preview to preview the source in the 3D view.
4. Click OK.
   - The ambient source appears in the specification tree.
   - The ambient source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.
You can find some .hdr files:

- Light Probe Image Gallery (free) (http://www.debevec.org/Probes/)
- Dosch Design (http://www.doschdesign.com/products/hdri/?sid=cb83c8f4fdd2c07b38794d75a4be8c8f)
- Sachform Technology (http://www.sachform.de/index.php?option=com_content&view=article&id=5&Itemid=4&lang=en)

Note that RTR Environment and PHS Environment types are not yet available.

HDRI has only relative luminance value not absolute. If you set the luminance to 1000 cd/m² then all pixels with the (1, 1, 1) color value will have 1000 cd/m².

The other colors luminance are defined relatively to this one.

HDRI are used to create nice renderings, not reliable ones.

### Parameters of an Ambient Source with Environment Type

**Zenith**

To set the zenith orientation, you must click a line in the graphics area.

You can click Reverse direction to reverse the direction.

**Properties**

- To set the north direction, click a line in the graphics area.
  
  You can click Reverse direction to reverse the direction.

- In the Luminance box, you must type or edit a luminance value.
  
  This value usually varies in the 1000 - 20000 cd/m² range.
  
  Pixels having (1, 1, 1) as values have the luminance defined in the panel.
  
  (1,1,1,) means HDRI's RGB value.
  
  This (1, 1, 1) is the floating point representation of the reference white color corresponding to the user defined luminance of the environment map.

- In the Spectrum boxes, you must browse a .spectrum file for each primary color.
  
  You also can edit the files by clicking Edit....

This can be created with the spectrum generator tool of the Spectrum Editor or download from the OPTIS Online Library (http://www.optis-world.com/download_software_libraries.asp).

- From the Environment type list, you must select a file or an environment.
  
  If you select Image File or HDRI File, you must browse an associated file.
  
  You also can edit the file by clicking Edit....
The ground plane is only available for ambient source with environment type in HDRI.
- In the Origin box, you must click a point in the graphics area.
The plane is defined by a point and a normal, this one is automatically the zenith direction.
- You must set a value in the Height spinbox.
The height corresponds to the height of the view compared to the plan.

**Ambient Source Visualization**

**Setting the Ambient Source as Background**

With Set Ambient Source as Background, you can visualize an ambient source in real time as it could be in simulation.
The visualization is made through the display of the ambient source with the type in the background of the CATIA window.
The orientation of the environment is related to the camera controlled by the user and the result is a real time.
Set Ambient Source as Background is available from the CATIA R19 release.

*To use Set Ambient Source as Background, you must click Tools, Options..., General, Display, Performance, and in the Miscellaneous section you must select the Enable OpenGL Shader check box.*

*You must open a product including an ambient source.*

When launching complex operations as optimization, it is recommended to disable the Ambient Source Visualization to avoid a slow down of the performances.

1. Click Set Ambient Source as background.
2. From the specification tree, in Sources section, select the ambient source to display in the 3D view background.
Natural light ambient source result

By clicking again Set Ambient Source as background, you disable the tool. If you modify the ambient source definition or the model, an automatic update of the visualization occurs.

Changing Background Exposures

- To make the background brighter, click Increment background exposure.
- To make the background darker, click Decrement background exposure.

Each click increments or decrements the level with a 15% factor. If you modify the ambient source definition or the model, an automatic update of the visualization occurs.

Ray File Source

Creating a Ray File Source

With a ray file source, you can use the OPTIS native ray file format to describe the emission of a light source. A ray file source is a pre-calculated source which emits all the rays in a .ray file. This file contains positions, directions, and wavelengths for each ray. You can create as many ray file sources as you want.

Sometimes, it is useful, to save the simulation time, to split a simulation in two parts. The first simulation can be dedicated to simulate the light propagation in parts with a definitive design (for instance the filament, the bulb and the socket of a lamp). The second simulation can be dedicated to simulate the light propagation in parts currently in the design process (for instance a reflector).
A ray file source can be created by using a ray file generated by the first simulation. Then, the ray file source can be used to replace the first part of the optical system in the second simulation. At each simulation done to optimize the second part of the optical system, the simulation time dedicated to the ray propagation in the first part is saved. Generally, with this tip, you can save between 20% and 80% of the simulation time.

1. Click Ray File Source (Sources).
2. Set the parameters (see page 50).
   - You can click Preview to preview the source in the 3D view.
3. Click OK.
   - The ray file source appears in the specification tree.
   - The ray file source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

![Ray file source](image)

**Parameters of a Ray File Source**

**Axis System**

To select one point in the Origin box, and two lines in the X Direction box and in the Y Direction box, you must click them in the graphics area.

You can click Reverse Direction to reverse the direction.

**Flux**

In the Flux group box, you can see the default value and unit of the flux.

- In the Unit group box you can modify the flux unit.
- Click From Ray File to edit the flux value and type a new value in the Flux box.

   If you define an old ray file that does not contain values in lumen, you cannot change the flux unit.
   If you need to use an old ray file, you can convert it to a more recent file format with the Ray File Editor to get values in lumen.

**Photometry**

In the Photometry group box:

- you can browse a .ray file that describes the light emission.
- you can click Edit to directly edit a selected ray file.

These ray files can be generated during a direct simulation by a rays map sensors, with another compatible software or given by some suppliers.

Note that the size of a ray file is roughly 30 MB per 1 Mrays. Free space disk consideration should be taken into account.

**Geometries**

In the Exit Geometries box, you can associate geometries to a ray file source.

You can select the different following elements:
Directly click them in the graphics area, or use the Other Selection option (see page 11).

The rays are propagated without interaction with associated geometries up to other external geometries. After an interaction with an external geometry, the rays are propagated taking into account the associated geometry.

With this functionality, you can, for example, use a ray file to describe the light emission of a lamp and in the same time to model the shadow created by the lamp geometry. Itself can be very critical in many optical systems.

**Thermic Surface Source**

**Creating a Thermic Surface Source**

A thermic surface can define a source for which the total flux and the spectrum are defined by the source's temperature and the optical properties of the support geometry.

A thermic surface source can be defined on any face of the geometry.

1. Click Thermic Surface Source (Sources).
2. Set the parameters (see page 51).
   - You can click Preview to preview the source in the 3D view.
3. Click OK.
   - The thermic surface source appears in the specification tree.
   - The thermic surface source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.
   - You can edit the thermic source properties (see page 58).

**Parameters of a Thermic Source**

**Flux**

In the Flux group box, you can see the unit and the value of the flux.

The flux is determined by calculating the emittance's integral on the geometry of the source.

**Intensity**

In the Intensity group box, you must select the type of intensity distribution.

The intensity diagram of a source describes in which directions is made the emission.
You can click Reverse direction to reverse the direction.

**Lambertian**

The simplest model is Lambertian which is a distribution law given by $\cos(\theta)$.

The intensity formula for Lambertian is $I = \cos(\theta)$

**Cos**

If you select Cos, in the N box, you can type or edit a value.

Luminance of a source with the Lambertian law, the $\cos\theta$ law where $N=10$ and the $\cos\theta$ law where $N=3$ for an intensity distribution.

**Emittance**

In SPEOS, emittance is calculated by averaging the wavelength range of the black body spectrum corresponding to a specific temperature.

In the Emittance group box, you can select a type from the Type list.

**Temperature Field**

If you select Temperature field, you can edit the axis system, the temperature field and the surface optical properties.

- In the Axis System group box, you can choose the origin, X and Y directions by clicking on the graphics area.
  
  You can click Reverse direction to reverse the direction.

- In the Temperature Field box, you can browse a .OPTTemperatureField file.

  .OPTTemperatureField file format includes description line, number of summits (Ns), number of triangles (Nt), coordinates $x,y,z$ of summits ($x$ Ns), coordinates $l,m,n$ of normals ($x$ Ns), index of summits of each triangle ($x$ Nt), temperature of each triangle ($x$ Nt).

- In the Surface optical properties (SOP), you can select a type from the Type list.
  
  - If you select Mirror, in the Reflectance box you can type or edit the reflectance value.
• If you select Library, you must browse a .simplescattering file. You also can edit a file by clicking Edit.

Temperature field thermic surface source

Emissive Face
If you select Emissive faces, you can edit the temperature and select several emissive faces.
• In the Temperature box, you can type or edit the value.
• In the Emissive faces box, you can select the following emissive elements in the graphics area.

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11). You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.
You must click the first element to start the selection, move the cursor over every needed face, and then click the last element when the selection is complete.
If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.
Emissive faces thermic surface source

Result for temperature field thermic surface source (tank) and 3 emissive faces (soldiers) at 150m

The emissivity calculation method is: \( \frac{\text{ThermalRadiance(Object)}}{\text{ThermalRadiance(PerfectBlackbody)}} \).

For bodies whose transmission, reflexion and absorption properties do not depend on wavelength, emissivity is calculated on a gray body model. It is constant which is in the interval \([0 ; 1]\) according to the absorption coefficient.

1 corresponds to the perfect blackbody. A blackbody is an object which does not reflect any electromagnetic radiations at any wavelength. The radiance from a blackbody is the radiance from a system in thermodynamic balance, defined by its temperature. The energy emitted by a blackbody is defined by its temperature and its emissivity. Its emittance is uniform. 0 corresponds to the perfect white body. A white body is an object which反射s all incident rays in all directions, in a uniform way. The gray body is thus a source with an emissivity lower than the blackbody, independent of frequency.

The emissivity calculation only takes the absorption at a normal incidence (0°) into account and does no vary according to wavelength or incidence.

The calculation of the absorption for opaque surface only relies on the principle that emissivity is equal to absorption.

Interactive Source

Creating an Interactive Source

An interactive source generates specific light rays which are useful to understand the behavior of a light beam through an optical system.

These specific light rays are used in interactive simulations and their purpose is not to model the real emission of the light source as a LED or a filament.

The interactive sources are generally created to be used in an interactive simulation.

1. Click Interactive Source \( \text{Interactive Source} \) (Sources).
2. Set the parameters (see page 55).
3. Click OK.

The interactive source appears in the specification tree.

The interactive source is displayed in the 3D view.
You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.

**Interactive source**

**Parameters of an Interactive Source**

To create an Interactive Source you must have two geometries, one for the propagation's start and one for the propagation's end.

For example, if you select Point-Direction from the Interactive Source Definition list, it means that the beginning geometry is a point and the end geometry a direction.

The Interactive Source makes the link between both geometries in a points to points way.

Then the propagation of the source is done through the system.

**Type**

In Type box, you must select the type of the source.

With the different types of interactive source, you can create any set of rays.

**Wavelength**

In Wavelength box, you can type or edit the wavelength value.

**Start**

In Start group box, you must select the start of the geometry by clicking it in the graphics area or in the specification tree.

In Sampling box, you can type or edit the sampling that you want for your geometry.

**End**

In End group box, you must select the end of the geometry by clicking it in the graphics area or in the specification tree.

In Sampling box, you can type or edit the sampling that you want for your geometry.
In case of direction, you can click Reverse direction to reverse the direction.

**Luminaire Source**

**Creating a Luminaire Source**

With Luminaire Source, you can model artificial light for outdoor and indoor lighting.

1. Click Luminaire Source (Sources).
2. Set the parameters (see page 56).
   - You can click Preview to preview the source in the 3D view.
3. Click OK.
   - The luminaire source appears in the specification tree.
   - The luminaire source appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source.
   - You can edit the luminaire source properties (see page 58).

![Luminaire source](image)

**Parameters of a Luminaire Source**

**Axis System**
To select one point in the Origin box, and two lines in the X Direction box and in the Y Direction box, you must click them in the graphics area.
You can click Reverse direction to reverse the direction.

**Intensity**
In the Intensity group box, you can browse a .ies or .ldt file that describes the light emission.
You also can edit a file by clicking Edit…. For more details, you can view IESNA LM-63 viewer or Eulumdat viewer.

**Spectrum**
In the Spectrum group box, you can select the spectrum type from the Type list.
- If you select Blackbody type, in the Temperature box, you can type or edit a value.
- If you select Library type, you must browse or edit a .spectrum file.

**Creating a Source Group**

With the Source Group, you can group several sources in the same group.
Source groups can only be used in a direct or inverse simulation and must be in the same product than the simulation.
You can add all source types excepted interactive and ambient sources.

All sources of a same group are stored in the same layer of a result file.
In some inverse simulations with a large number of sources as for a cockpit application, with Source group you can join several displays in the same layer in order to save memory when Separate data by layer / source parameter of the sensor is activated.
1. Click Source Group (Sources).
2. Click sources in the specification tree.
   Sources can be source groups.
3. Click OK.
   You can reframe on, hide, show, edit, copy, paste or delete (see page 57) the source group.
   You can edit the source group properties (see page 58).

Sources Management
Managing the Sources within the Specification Tree
All the created sources appear in the specification tree.
You can edit source definition by double-clicking on the source.
You can right-click on the source to manage the source.
- If you want to focus on a source in the specification tree, click Center graph.
- If you want to focus on a source in the graphics area, click Reframe On.
- If you want to hide or display the source, click Hide/Show.
- If you want to edit the source properties, click Properties.
  For more details, you can view Properties (see page 58).
- If you want to cut, copy or paste the source, click Cut, Copy, Paste or Paste Special....
- If you want to move the source, you can drag and drop it from a set to another.
- If you want to delete the source, click Delete.

Managing the Sub-Trees
- To open a source sub-tree from the specification tree, you can double-click on a source, or right-click on a source and click Open Sub-Tree.
- From surface source sub-tree, you can double-click on source parameters to set a value in the corresponding spinbox.
  Note that the Number of rays parameter is used for visualization.
  If the number of rays is equal to zero, the rays display is stopped.

- From ray file, thermic, luminaire and display sources sub-tree, you can double-click on source parameters to set a value in the corresponding spinbox.
  For Filename, you can type the file name.
- From interactive and ambient sources sub-trees, you can double-click on sources parameters to set a value in the corresponding spinbox.

Managing the Source Objects
By right-clicking on a source from the specification tree, you can click Source Name object.
- If you want to edit the source definition, click Definition....
• If you want to gather several source objects into a same sub-group, click Create Set (see page 11).
• If you want to update a source, click Local Update.
• If you want to update a series of sources without blocking the process if one of them leads to an error, click Local Update Without Notification.
• If you want to activate or deactivate a component, click Activate / Deactivate Component.
• If you want to move up and down the source in the Sources section of the specification tree, click Move up or Move down.

Properties
To edit the sources properties, you can:
1. Select a source in the specification tree or in the 3D view.
2. Click Edit SPEOS properties.
   The properties of the source open.
-Or-
1. Right-click a source in the specification tree.
2. Click Properties.
3. Click More >.
4. Edit the SPEOS properties of the source in the corresponding tabs.

Setting the IES Visualization Properties for a Surface Source
You must click More… to edit the IES visualization tab.
• You can select the Display intensity diagram check box to display the intensity diagram.
  • The orientation depends on the selected axis.
  • The position is at the barycenter of the selected faces.
  • The size of the display corresponds to the radius of the enclosing sphere.
  • The color depends on the spectrum's wavelength of the source.
  The display of the intensity diagram in the 3D view is automatically done for the Library type.

Display of the intensity diagram of a surface source
• You can select the Fixed size check box to set a length value in the spinbox.
  With fixed size, you can specify a normalized size for the intensity diagram.
Setting the IES Visualization Properties for a Luminaire Source

You must click More… to edit the IES visualization tab.

- You can select the Display intensity diagram check box to display the intensity diagram.
  - The orientation and the position depend on the axis system of the source.
  - The size of the display corresponds to the radius of the enclosing sphere of the geometry contained in the IES definition.
  - The color depends on the spectrum's wavelength of the source.

Display of the intensity diagram of a luminaire source

- You can select the Fixed size check box to set a length value in the spinbox.
  With fixed size, you can specify a normalized size for the intensity diagram.

Setting the Thermic Source Parameters

Parameters tab is available for Thermic Surface Source.
You must click More… to edit the Parameters tab.

- You can select the Display temperature field mesh check box.
  To improve the display performances in case of complex geometries, you can click to clear the check box.
- You can select the Automatic levels check box to manage the temperature levels for the scale of wrong colors.
  In automatic mode, minimal and maximal temperatures are automatically calculated.
  In manual mode, you must set the values in the spinboxes.

Sensors

Irradiance Sensor

Creating an Irradiance Sensor

With an Irradiance Sensor, you can compute the illuminance in Lux or the irradiance in Watt/m².

1. Click Irradiance Sensor (Sensors).
2. Set the parameters (see page 60).
3. Click OK.
   The irradiance sensor appears in the specification tree.
   The irradiance sensor appears in the 3D view.
   Note that you can change sensor's dimensions and directions from the 3D view.
You can reframe on, hide, show, edit, copy, paste or delete (see page 101) the sensor. You can edit the sensor properties (see page 101).

Irradiance sensor

Parameters of an Irradiance Sensor

Type
You must select the type from the Type list.

- Click Photometric to get results in Lux.
- Click Colorimetric to get results both in Lux and Watt/m².
- Click Radiometric to get results in Watt/m².

The Colorimetric option is available only with a Colorimetric license.

Ray File
You can activate the Ray file.
If you do it, you can create a ray file containing all rays passing through the sensor during the propagation.
The ray file is available at the end of the simulation under the result associated to the sensor.
This ray file can be used in a Ray File Source (see page 49) in another simulation. -
Axis System
With the axis system parameters, you can define the orientation of the sensor.
When you select the sensor in the specification tree, the Z axis is displayed in the 3D view.
The Z axis is a normal axis to the detector's surface which defines the sensitive face of the detector.
The photons are integrated when they arrive there.
You also have different types of integration to measure illuminance. Illuminance is the physical quantity defined by the luminous flux received on sensor area.
Depending on the simulation you need, you can use different integration types.
The most frequent type is the planar type. In this type, the illuminance on a point is calculated by the cosine on the angle of incidence $\epsilon$. The formula is:

$$E = I_0 \frac{\cos \epsilon}{D^2}$$

For the other types, calculation is based on standard EN-13201 (http://www.e-streetlight.com/standard_en13201.htm), which gives mathematical formulas equivalent to different types of illuminance.
Compared to the EN-13201 standard, several parameters are simplified.
1. Select one point in the Origin box, and two lines in the X direction box and in the Y direction box, you must click them in the graphics area.
   You can click Reverse direction to reverse the direction.
2. Select an Integration type and then click a direction in the graphics area to select it in the Integration direction box if necessary.
This direction can be selected from the geometry.

- You can select Planar (see page 63).
  In this case, defining an integration direction is optional.
- You can select Radial (see page 64).
  In this case, no integration direction is required.
- You can select Hemispherical (see page 65).
  In this case, defining an integration direction is mandatory.
- You can select Cylindrical (see page 65).
  In this case, no integration direction is required.
- You can select Semi-Cylindrical (see page 66).
  In this case, you must select a direction parallel to the sensor plan.

Default type is Planar.
You can click Reverse direction to reverse the direction.

![Irradiance sensor with an oblique integration direction](image)

With the axis system parameters, you can define the orientation of the sensor.

**X and Y**
In the X and Y group boxes, you can type or edit the start, end and sampling values.

The sampling is used to define the number of pixels of the XMP map.
You can select the Mirrored Extent check box to have a sensor symmetrical compared to the selected axis system.

**Wavelength**
Wavelength is only available for the Colorimetric type.
400 and 700 nanometers are the lower and higher borders for the spectral sampling.
Every wavelength beyond these borders are not taken into account by the sensor.
When sampling the spectrum, you calculate with the FWHM instead of the peak value for the according wavelength.
It is dangerous to have a higher sampling of the source versus the spectrum.

**Layer**
If you click More>>>, the Layer list appears.
You can define if all photometric results are stored in the same XMP layer or not.
In case of large XMP sampling with a lot of sources as the size of required memory could be critical, you must click None.

- If you select Source from the list, the result includes one layer per active source.
  Sources power or spectrum can then be changed using virtual lighting controller present in Virtual Photometric Lab and Virtual Human Vision Lab.
- If you select Face from the list, the result includes one layer per face selected using the surface contribution analyzer. You can select several faces in case of a surface or volume.
  You can select the different following elements from the graphics area:
Features

Page 63 of 251

For more details, you can view Surface Contribution Analyzer (see page 66).

- If you select Polarization from the list, the result includes one layer per Stokes parameter using the polarization option.

  Stokes parameters are displayed using the layers of the Virtual Photometric Lab.
  For more information, you can view Stokes Parameters.
  The Polarization parameter is only available if you selected the Radiometric type.

Inverse Simulation Optimization

You can click More>> to view the Output faces for inverse simulation optimization option.

Select the specific different following elements to limit the inverse propagation and reduce the inverse simulation calculations costs:

For more details, you can view Surface Contribution Analyzer (see page 66).

- If you select Polarization from the list, the result includes one layer per Stokes parameter using the polarization option.

  Stokes parameters are displayed using the layers of the Virtual Photometric Lab.
  For more information, you can view Stokes Parameters.
  The Polarization parameter is only available if you selected the Radiometric type.

Inverse Simulation Optimization

You can click More>> to view the Output faces for inverse simulation optimization option.

Select the specific different following elements to limit the inverse propagation and reduce the inverse simulation calculations costs:

Select the specific different following elements to limit the inverse propagation and reduce the inverse simulation calculations costs:

Select the specific different following elements to limit the inverse propagation and reduce the inverse simulation calculations costs:

You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.

If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

Planar Illuminance

On the following example, the source is simulated by a local point source. The source can also be an extended light source like luminaire, ambient, surface source, etc.

Note that in this integration type, the pixel is only sensitive on one side. Its sensitivity is lambertian.

Three types of integration direction can be simulated for Planar illuminance.

Horizontal plan

The horizontal illuminance is the most common way to calculate illuminance. The integration direction is perpendicular to the horizontal plan and the surface sensor. The normal illuminance follows the Bouguer law.

Vertical plan

When the surface sensor is applied vertically, the lateral orientation becomes a significant parameter to determine the illuminance. The integration direction is perpendicular to the vertical plan and parallel to the surface sensor. An example of this can be a wall on the road.
Illuminance on vertical sensor with lateral deviation (angle $\alpha$)

In the specific case where $\alpha$ is equal to 0 (like the object on the right of the above figure), the illuminance calculation is the same as for the horizontal type. It does not depend on the $\alpha$ factor. The mechanical plan only is different, so the two coordinates systems have different orientations.

$$E_{\text{vertical specific}} = \frac{I_s \sin \varepsilon}{D_2^2}$$

General case

In the general case you must define the integration direction. The same integration direction is applied on each pixel of the sensor. On the figure below, this direction is perpendicular to blue mechanical plans.

Planar illuminance in general case

Radial Illuminance

On the following example, the source is simulated by a local point source. The source can also be an extended light source like luminaire, ambient, surface source, etc.

Note that in this integration type, the sensitivity of the pixel does not depend on where the rays are coming from. The integration direction is the incident flux. This direction is on the vertical plan at right-angle to the surface. Then, the angle of incident $\varepsilon$ is equal to 0° and $\cos \varepsilon = 1$. The illuminance formula is:
Hemispherical Illuminance

On the following example, the source is simulated by a local point source. The source can also be an extended light source like luminaire, ambient, surface source, etc.

Note that in this integration type, the sensor is sensible to light incoming from all directions except the direction exactly opposed to the integration direction.

The hemispherical illuminance is an addition of horizontal and radial illuminance. The integration direction is perpendicular to the sensor plane. The integration direction is the same like horizontal illuminance (perpendicular to the horizontal plan).

\[ E_{\text{Hemispherical}} = \frac{I_\varepsilon(1 + \cos \varepsilon)}{4D^2} \]

Cylindrical Illuminance

On the following example, the source is simulated by a local point source. The source can also be an extended light source like luminaire, ambient, surface source, etc.

Note that in this integration type, the sensor is sensible to light incoming from all directions except the direction exactly normal to the sensor plane.

The cylindrical illuminance can be defined by the specific case of vertical illuminance (when \( \alpha = 0^\circ \)). Because of the rotational symmetry (around z axis) only the angle \( \varepsilon \) is important, we do not need a specific integration direction.
Semi-Cylindrical Illuminance

On the following example, the source is simulated by a local point source. The source can also be an extended light source like luminaire, ambient, surface source, etc.

Note that in this integration type, the sensor is sensitive to light incoming from all directions, except the directions included in a half plan delimited by the cylinder axis and situated behind the half cylinder.

Contrary to the cylindrical illuminance, we need an integration direction to calculate the semi-cylindrical illuminance. In addition, the illuminance depends on the lateral deviation (like the vertical illuminance).

Surface Contribution Analyzer

You must have created a direct or an inverse simulation.

With Surface Contribution Analyzer, you can generate a XMP map with one layer for each face selected for the analysis.

Then using the Virtual Photometric Lab, you can display and measure the contribution of each face on the sensor.
This tool is useful for the reflector analysis. For example, if you have a system included a reflector and a lamp:

With a detector, you can get this type of result which gives you the illuminance of your system on the sensor:

The surface contribution analyzer gives the same result but you can have the contribution of each surface that you have selected for the analysis.

Using Surface Contribution Analyzer

With Surface Contribution Analyzer, you can visualize the contribution of each photon that has been transmitted or reflected on the selected surfaces before reaching the sensor.

1. From the Irradiance Sensor Definition dialog box, click More>>, and then select Face from the Layer list.
2. In the Face filtering box, you must click the faces in the graphics area.
   You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.
   You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.
If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection....

Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

3. Set the parameters (see page 68).

4. Update the simulation.

At the end of it, you get the XMP map.

The result is the same as previously except that the XMP map has one layer for each selected surface. The name of each layer is given by the name of the face and the name of the component.

Note that you can select a layer in the list to access to the contribution of this surface.

Parameters of Surface Contribution Analyzer

From the Filtering mode list, you must select Last Impact or Intersected one time.
- If you select Last Impact, the ray is integrated in the layer corresponding to a surface if the surface is the last one the ray intersects before hitting the sensor. Otherwise, this ray’s contribution is integrated in the All other faces layer.
If you select Intersected one time, the ray is integrated in the layer corresponding to a surface if:
  - the ray intersects the surface at least one time
• the last surface the ray hits before the sensor is not selected in the Surfaces list.
If 1 is respected but not 2, the ray is integrated in the layer corresponding to the last surface.
Otherwise, the ray's contribution is integrated in the All other faces layer.

**Intensity Sensor**

**Creating an Intensity Sensor**

With an Intensity Sensor, you can compute radiant intensity in Watt/sr and luminous intensity in Candela.

1. Click **Intensity Sensor** (Sensors).
2. Set the parameters (see page 73).
3. Click OK.

The intensity sensor appears in the specification tree.

The intensity sensor appears in the 3D view.

Note that you can change sensor's dimensions and directions from the 3D view.

You can reframe on, hide, show, edit, copy, paste or delete (see page 101) the sensor.

You can edit the sensor properties (see page 101).

---

**Parameters of an Intensity Sensor**

**Type**

You must select the type from the Type list.

- Click **Photometric** to get results in Candela.
- Click **Colorimetric** to get results both in Candela or Watt/m².
- Click **Radiometric** to get results in Watt/sr.

The Colorimetric option is available only with a Colorimetric license.

**Format**

You must select the format from the Format list.


For more details about the Eulumdat format, you can view Data Format for Exchange of Luminaire Data (http://www.helios32.com/Eulumdat.htm).

**Near Field**

This feature is useful only to compare simulated intensity to measured intensity on small size devices. You cannot use the result of a near field sensor to model the near field of a light source for instance.

The result of a simulation using a near field sensor is closer to physical experiments.
- If you do not select the Near field box, the intensity is located at the infinite.
- If you select the Near field box, you must set a value in the Cell distance and Cell diameter spinboxes to match the measurement tool.
  - The size of the cell must be superior to one pixel.
  - Sensor visualization in the 3D view is set according to the cell distance value.
  - The results obtained with a near field sensor can be inaccurate on the edge of the map, over a width equal the radius of a cell.

**Axis System**
- If you selected the XMP format, you must select one point in the Origin box, and two lines in the X Direction box and in the Y Direction box, by clicking them in the graphics area.
- If you selected the IESNA A, C, the Eulumdat or the OPTIS format, you must select one point in the Origin box, and two lines in the Polar axis box and in the H 0 axis box, by clicking them in the graphics area.
- If you selected the IESNA B format, you must select one point in the Origin box, and two lines in the Polar axis box and in the V 0 axis box, by clicking them in the graphics area.

You can click Reverse direction to reverse the direction.

With the axis system parameters, you can define the orientation of the sensor. 
The three points give two vectors X and Y.

**Orientation**
- Orientation is only available for the XMP format.
- From the Orientation list, you must select the orientation.
- When selecting Conoscopic, in the Size group box, you must type or edit the theta max and the sampling values.

![Meridian/Parallel orientation](image1)
![Conoscopic orientation](image2)

**Meridian/Parallel orientation**
**Conoscopic orientation**

**X and Y**
- In the X and Y group boxes, you must type or edit the start, end and sampling values.

You can select the Mirrored Extent check box to have light from all space.

**H Plane and V Plane**
- For IESNA, Eulumdat or OPTIS format, you must adjust the sampling value in the H plane and V plane group boxes by typing or editing it.

**Adaptive Sampling**
- You can browse a .txt file in the Adaptive sampling box.
- Format of the file for IES A and B is the following.
  - Line 1 is a header.
    - OPTIS - Intensity Distribution sampling file v1.0
  - Line 2 is a comment.
    - Example for IES B
- Line 3 contains following values.
  HSamplingNumber -90 Angle1 Angle2... AngleN 90
- Line 4 contains following values.
  VSamplingNumber -90 Angle1 Angle2 ... AngleN 90

HSamplingNumber corresponds to the number of samples on H plane.
VSamplingNumber corresponds to the number of samples on V plane.

When the sampling goes from -90 and 90 degrees for an IES A, the sample list has to begin at -90 and finish at 90.

Format of the file for IES C, Eumldat and Optis Intensity is the following.
- Line 1 is a header.
  OPTIS - Intensity Distribution sampling file v1.0
- Line 2 is a comment.
  Example for IES C, Eumldat and Optis Intensity (Radio and Photo)
- Line 3 contains following values.
  HSamplingNumber 0 Angle1 Angle2… AngleN 360
- Line 4 contains following values.
  VSamplingNumber 0 Angle1 Angle2 ... AngleN 180

HSamplingNumber corresponds to the number of samples on H plane.
VSamplingNumber corresponds to the number of samples on V plane.

Examples of adaptive sampling file are located in ..\OPTIS\Standards\Photometry\Intensity_Distribution directory.

If the simulation does not launch, you must check the adaptive sampling file.

Layer

When using the XMP format, you can define if all photometric results are stored in the same XMP layer or not.

In case of large XMP sampling with a lot of sources as the size of required memory could be critical, you must click None.

By clicking More>>, the Layer list appears.
- If you select Source from the list, the result includes one layer per active source.
  Sources' power or spectrum can then be changed using virtual lighting controller present in Virtual Photometric Lab and Virtual Human Vision Lab.
- If you select Face from the list, the result includes one layer per face selected using the surface contribution analyzer. You can select several faces in case of a surface or volume.
  You can select the specific different following elements:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
<tbody>
<tr>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
</tbody>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11). Click faces from the graphics area to add them to the Face filtering list.

You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.
If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

For more details, you can view Surface Contribution Analyzer. (see page 75)

In the Design Table box, you can select a design table for multi-configuration.

Surface Contribution Analyzer

You must have created a direct or an inverse simulation.

With Surface Contribution Analyzer, you can generate a XMP map with one layer for each face selected for the analysis.
Then using the Virtual Photometric Lab, you can display and measure the contribution of each face on the sensor.
This tool is useful for the reflector analysis.
For example, if you have a system included a reflector and a lamp:

With a detector, you can get this type of result which gives you the illuminance of your system on the sensor:

The surface contribution analyzer gives the same result but you can have the contribution of each surface that you have selected for the analysis.

Using Surface Contribution Analyzer

With Surface Contribution Analyzer you can visualize the contribution of each photon that has been transmitted or reflected on the selected surfaces before reaching the sensor.

1. From the Intensity Sensor Definition dialog box, click More>>, and then select Face from the Layer list.
2. In the Face filtering box, you must click the faces in the graphics area.

Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection....
Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

3. Set the parameters (see page 77).

4. Update the simulation.

At the end of it, you get the XMP map.

The result is the same as previously except that the XMP map has one layer for each selected surface. The name of each layer is given by the name of the face and the name of the component.

Note that you can select a layer in the list to access to the contribution of this surface.

Note that you can also display the Virtual Lighting Controller which allows switching on or off the surfaces you want.

**Parameters of Surface Contribution Analyzer**

From the Filtering mode list, you must select Last Impact or Intersected one time.
- If you select Last Impact, the ray is integrated in the layer corresponding to a surface if the surface is the last one the ray intersects before hitting the sensor. Otherwise, this ray's contribution is integrated in the All other faces layer.
- If you select Intersected one time, the ray is integrated in the layer corresponding to a surface if:
  - the ray intersects the surface at least one time
- the last surface the ray hits before the sensor is not selected in the Surfaces list.
If 1 is respected but not 2, the ray is integrated in the layer corresponding to the last surface.
Otherwise, the ray’s contribution is integrated in the All other faces layer.

**Radiance Sensor**

**Creating a Radiance Sensor**

With a Radiance Sensor, you can compute radiance in Watt/sr/m² and luminance in Candela per square meters.

1. Click Radiance Sensor (Sensors).
2. Set the parameters (see page 82).
3. Click OK.

   The radiance sensor appears in the specification tree.
   The radiance sensor appears in the 3D view.

   Note that you can change sensor’s dimensions and directions from the 3D view.
   You can reframe on, hide, show, edit, copy, paste or delete (see page 101) the sensor.
   You can edit the sensor properties (see page 101).

**Parameters of a Radiance Sensor**

**Type**

You can select the type from the Type list.

- Click Photometric to get results in Candela per square meters.
- Click Colorimetric to get results in Candela per square meters or Watt/sr/m².
- Click Radiometric to get results in Watt/sr/m².

The Colorimetric option is available only with a Colorimetric license.

**Definition From**

You can select the definition from the Definition from list.

**Point Line and Dimension**

- You must adjust the Axis System parameters.
  To select one point in the Origin box, and two lines in the X Direction and Y Direction boxes, you must click them in the graphics area.
  You can click Reverse direction to reverse the direction.
With the axis system parameters, you can define the orientation of the sensor. The three points give two vectors X and Y.

- In the X and Y group boxes, you can type or edit the start, end and sampling values.
- You can select the Mirrored Extent check box to have light from all space.
- You must select the type from the Observer Type list.
  - If you select Focal, you can type or edit a length value in the Focal box.
  - Note that you can select the Automatic framing check box or click Automatic Framing on Feature.
  - For more details, you can view Using the Automatic Framing on a Radiance Sensor (see page 84).
  - If you select Observer, you must click an observer in the graphics area in the Observer box.

---

**Focal type**

**Observer type**

**Camera**

- In the Camera box, you must click a camera in the graphics area.
- You must set the value in the Camera H/V Ratio spinbox.
- In the X and Y group boxes, you must set the start, end and sampling values in the spinboxes.

**Manikin**

- In the Manikin box, you must click a manikin in the graphics area.
- In the X and Y group boxes, you must set the sampling value in the spinboxes.
Wavelength

Wavelength is only available for the Colorimetric type. 400 and 700 nanometers are the lower and higher borders for the spectral sampling. Every wavelength beyond these borders are not taken into account by the sensor. When sampling the spectrum, you calculate with the FWHM instead of the peak value for the according wavelength. It is dangerous to have a higher sampling of the source versus the spectrum.

Layer

By clicking More>>, the Layer list appears. You can define if all photometric results are stored in the same XMP layer or not. In case of large XMP sampling with a lot of sources as the size of required memory could be critical, you must click None.

- If you select Source from the list, the result includes one layer per active source. Sources' power or spectrum can then be changed using virtual lighting controller present in Virtual Photometric Lab and Virtual Human Vision Lab.
- If you select Face from the list, the result includes one layer per face selected using the surface contribution analyzer. You can select several faces in case of a surface or volume.

For more details, you can view Surface Contribution Analyzer (see page 85).

You can select the different following elements:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
<tbody>
<tr>
<td>☑️</td>
<td>☑️</td>
<td>☑️</td>
</tr>
</tbody>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11).

You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.

If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

In the Design Table box, you can select a design table for multi-configuration.

Using the Automatic Framing on a Radiance Sensor

With Automatic framing on feature, you can reframe the camera on a radiance sensor.
This corresponds to the Automatic framing on feature option with the fact that you can then again manipulate the camera.

For more details you can view Parameters of a Radiance Sensor (see page 82).

1. Click Automatic framing on feature (Visualization tools).
2. Select a radiance sensor in the specification tree.

---

**Surface Contribution Analyser**

*You must have created a direct or an inverse simulation.*

With Surface Contribution Analyzer, you can generate a XMP map with one layer for each face selected for the analysis.

Then using the Virtual Photometric Lab, you can display and measure the contribution of each face on the sensor.
This tool is useful for the reflector analysis. For example, if you have a system included a reflector and a lamp:

With a detector, you can get this type of result which gives you the illuminance of your system on the sensor:

The surface contribution analyzer gives the same result but you can have the contribution of each surface that you have selected for the analysis.

**Using Surface Contribution Analyzer**

With Surface Contribution Analyzer you can visualize the contribution of each photon that has been transmitted or reflected on the selected surfaces before reaching the sensor.

1. From the Radiance Sensor Definition dialog box, click More>>, and then select Face from the Layer list.
2. In the Face filtering box, you must click the faces in the graphics area.

   Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection....
Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

3. Set the parameters. (see page 87)

4. Update the simulation.

At the end of it, you get the XMP map.

The result is the same as previously except that the XMP map has one layer for each selected surface. The name of each layer is given by the name of the face and the name of the component.

Note that you can select a layer in the list to access to the contribution of this surface.

Note that you can also display the Virtual Lighting Controller which allows switching on or off the surfaces you want.

### Parameters of Surface Contribution Analyzer

From the Filtering mode list, you must select Last Impact or Intersected one time.
• If you select Last Impact, the ray is integrated in the layer corresponding to a surface if the surface is the last one the ray intersects before hitting the sensor. Otherwise, this ray's contribution is integrated in the All other faces layer.
- If you select Intersected one time, the ray is integrated in the layer corresponding to a surface if:
  - the ray intersects the surface at least one time
- the last surface the ray hits before the sensor is not selected in the Surfaces list.
If 1 is respected but not 2, the ray is integrated in the layer corresponding to the last surface.
Otherwise, the ray’s contribution is integrated in the All other faces layer.

### 3D Irradiance Sensor

With a 3D Irradiance Sensor, you can compute irradiance in Watt/sr/m² or luminance in Lux on the geometry itself.

1. Click 3D Irradiance Sensor (Sensors).
2. In the Ray file box, select true or false from the list.
3. In the Faces box, you can select the different following elements:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
<tbody>
<tr>
<td>✔️</td>
<td>✔️</td>
<td>✗</td>
</tr>
</tbody>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11).

In the Bodies box, click the faces or the bodies in the graphics area.

You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.

If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

4. Click OK.

The 3D irradiance sensor appears in the specification tree.

The 3D irradiance sensor appears in the 3D view.

You can reframe on, hide, show, edit, copy, paste or delete (see page 101) the sensor.

Note that the meshing shown in the 3D view is defined in the Meshing panel. To define the final meshing of the sensor, you can view Setting the Simulation Properties (see page 138).
3D Energy Density Sensor

Creating a 3D Energy Density Sensor

With a 3D Energy Density Sensor, you can compute the volumic absorption in Lumen/m$^3$ or Watt/m$^3$.

1. Click 3D Energy Density Sensor (Sensors).
2. Set the parameters (see page 93).
3. Click OK.

   The 3D energy density sensor appears in the specification tree.
   The 3D energy density sensor appears in the 3D view.

You can reframe on, hide, show, edit, copy, paste or delete (see page 101) the sensor.

Parameters of a 3D Energy Density Sensor

Type
You can select the sensor type from the Type list.

- By clicking Photometric, you get results in Lumen/m$^3$.
- By clicking Radiometric, you get results in Watt/m$^3$.

Axis System
To select one point in the Origin box, and two lines in the X Direction box and in the Y Direction box, you must click them in the graphics area.

You can click Reverse direction to reverse the direction.

With the axis system parameters, you can define the orientation of the sensor.

The three points give two vectors X and Y.

Dimensions
In X, Y and Z Size boxes, you can type or edit the values.

Sampling
In X, Y and Z Sampling boxes, you can type or edit the values.

The sampling is used to define the number of pixels of the map.

In the Design Table, you can select a design table for multi-configuration.

Layer
By clicking More>>, the Layer list appears.

You can define if all photometric results are stored in the same XMP layer or not.
In case of large XMP sampling with a lot of sources as the size of required memory could be critical, you must click None.

- If you select Source from the list, the result includes one layer per active source. Sources' power or spectrum can then be changed using virtual lighting controller present in Virtual Photometric Lab and Virtual Human Vision Lab.
- If you select Face from the list, the result includes one layer per face selected using surface contribution analyzer. You can select several faces in case of a surface or volume.

For more details, you can view Surface Contribution Analyzer (see page 94). You can select the specific different following elements:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11). You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.

You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete. If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.

In the Design Table box, you can select a design table for multi-configuration.

**Surface Contribution Analyzer**

You must have created a direct or an inverse simulation.

With Surface Contribution Analyzer, you can generate a XMP map with one layer for each face selected for the analysis. Then using the Virtual Photometric Lab, you can display and measure the contribution of each face on the sensor. This tool is useful for the reflector analysis.

For example, if you have a system included a reflector and a lamp:
With a detector, you can get this type of result which gives you the illuminance of your system on the sensor:

![Image of illuminance result]

The surface contribution analyzer gives the same result but you can have the contribution of each surface that you have selected for the analysis.

**Using Surface Contribution Analyzer**

With Surface Contribution Analyzer you can visualize the contribution of each photon that has been transmitted or reflected on the selected surfaces before reaching the sensor.

1. From the 3D Energy Density Sensor Definition dialog box, click More>>, and then select Face from the Layer list.
2. In the Face filtering box, you must click the faces in the graphics area.
   
   Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection....
   
   Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

3. Set the parameters (see page 96).

4. Update the simulation.
   
   At the end of it, you get the XMP map.

   The result is the same as previously except that the XMP map has one layer for each selected surface. The name of each layer is given by the name of the face and the name of the component.
Note that you can select a layer in the list to access to the contribution of this surface.

Note that you can also display the Virtual Lighting Controller which allows switching on or off the surfaces you want.

Parameters of Surface Contribution Analyzer
From the Filtering mode list, you must select Last Impact or Intersected one time.
- If you select Last Impact, the ray is integrated in the layer corresponding to a surface if the surface is the last one the ray intersects before hitting the sensor. Otherwise, this ray's contribution is integrated in the All other faces layer.
- If you select Intersected one time, the ray is integrated in the layer corresponding to a surface if:
  - the ray intersects the surface at least one time
- the last surface the ray hits before the sensor is not selected in the Surfaces list.
If 1 is respected but not 2, the ray is integrated in the layer corresponding to the last surface.
Otherwise, the ray's contribution is integrated in the All other faces layer.

**Sensors Management**

**Managing the Sensors within the Specification Tree**

All the created sensors appear in the specification tree.

You can edit sensor definition by double-clicking on the sensor.

You can right-click on the sensor to manage the sensor.

- If you want to focus on a sensor in the specification tree, click Center graph.
- If you want to focus on a sensor in the graphic area, click Reframe On.
- If you want to hide or display the sensor, click Hide/Show.
- If you want to edit the sensor properties, click Properties.

For more details, you can view Sensor Properties (see page 101).

- If you want to cut, copy or paste the sensor, click Cut, Copy, Paste or Paste Special....
- If you want to move the sensor, you can drag and drop it from a set to another.
- If you want to delete the sensor, click Delete.

**Managing the Sensors Sub-Trees**

To open a sensor sub-tree from the specification tree, you can double-click on a sensor, or right-click on a sensor and click Open Sub-Tree.

- From sensor sub-tree, you can double-click on sensor parameters to set a value in the corresponding spinbox.
- From radiance sensor, you can double click Save Spectral Data, and then you must select true or false.
- From intensity sensor, you can double-click Adaptive Sampling, and then add a selection.

**Managing the Sensors Objects**

By right-clicking on a sensor from the specification tree, you can click Sensor Name object.

- If you want to edit the sensor definition, click Definition....
- If you want to gather several sensor objects into a same sub-group, click Create Set (see page 11).
- If you want to update a sensor, click Local Update.
- If you want to update a series of sensors without blocking the process if one of them leads to an error, click Local Update Without Notification.
- If you want to activate or deactivate a component, click Activate / Deactivate Component.
- If you want to move up and down the sensor in the Sensors section of the specification tree, click Move up or Move down.

**Sensors Properties**

To edit the sensors properties, you can:

1. Select a sensor in the specification tree or in the 3D view.
2. Click Edit SPEOS properties.
   The properties of the sensor open.

-Or-

1. Right-click a sensor in the specification tree.
2. Click Properties.
3. Click More >.
4. Edit the SPEOS properties of the sensor in the corresponding tabs.

**Setting the Radiance Sensor Parameters**

*You must click More... to edit the Parameters tab.*

- In the Integration angle (for Direct simulation only), you must type or edit the integration angle value. This parameter defines the half width of the full integration cone. Integration Angle default value is 5 degrees. For more details, you can view Integration Angle for a Direct Simulation.
- If you select the Display integration cone (for Visualization only) check box, you can display the integration angle in the 3D view.

![Radiance sensor with integration angle](image)

- You can activate the Save spectral data (for Inverse simulation only). If you deactivate it, the XMP results are reduced. It is useful when the amount of 32 bits memory is too low and with wavelength and X, Y sampling.

  Example of gain:
  Save spectral data activated corresponds to 17 906 KB.
  Save spectral data deactivated corresponds to 5 003 KB.

**Setting the Intensity Sensor Parameters**

- In the Integration angle (for Direct simulation only) box, you must type or edit the integration angle value. This parameter appears for IESNA and Eulumdat formats. For more details, you can view Integration Angle for a Direct Simulation. In case of near field intensity, the integration angle value is automatically calculated from the cell diameter value and the distance value. Integration angle = \( \tan^{-1} \left( \frac{\text{cell diameter}}{2} / \text{distance} \right) \)
- In the Radius (for Visualization only) box, you must type or edit the radius value. This parameter does not affect the simulation. In case of near field intensity, the radius value is automatically calculated from the cell distance value.
- You can activate the Flip XMP horizontally.

  This parameter appears only for XMP format. This horizontal flip parameter avoids end-users to manually flip the result at the end of each update.
Setting the Grid Properties

Grid tab is available for Irradiance Sensor, Intensity Sensor and Radiance Sensor.

With the Grid tab, you can display a grid on the sensor.

You must click More… to edit the Grid tab.

If you select true from the Show grid list, you can set the values in the Origin spinbox and in the Step spinbox.

The grid properties determined in this tab are applied by default when you open the XMP result of a simulation in the Virtual Photometric Lab.

Simulations

Simulations Compatibility

These are compatible sources and sensors with the different types of simulation.

<table>
<thead>
<tr>
<th>SIMULATIONS</th>
<th>SOURCES</th>
<th>SENSORS</th>
</tr>
</thead>
</table>
| Interactive Simulation | Surface Source (see page 35)  
Ray File Source (see page 49)  
Thermic Surface Source (see page 51)  
Interactive Source (see page 54)  
Luminaire Source (see page 56) | Irradiance Sensor (see page 59)  
Intensity Sensor (see page 73) |

Direct Simulation | Surface Source (see page 35)  
Ray File Source (see page 49)  
Thermic Surface Source (see page 51)  
Luminaire Source (see page 56) | Irradiance Sensor (see page 59)  
Intensity Sensor (see page 73)  
Radiance Sensor (see page 82)  
Immersive Sensor  
3D Irradiance Sensor (see page 92) |
### Interactive Simulation

The main goal of an Interactive Simulation is to display the propagation of rays in the 3D view in order to understand the behavior of a light beam in an optical system.

With an interactive simulation, you cannot measure a light quantity, you can only get a visual feedback of the propagation.

Thanks to the low number of rays required by this simulation, the result is synchronized with the associated geometries.

This functionality is a very useful tool to understand quickly how a design modification change the optical behavior.

### Creating an Interactive Simulation

With an Interactive Simulation, you can generate ray tracing to understand the optical system's qualitative behavior.

1. Click Interactive Simulation (Simulations).
2. Select Sources, Geometries and Sensors involved in this simulation.
   - If no geometry is associated to the interactive simulation, a body is always completely added to a simulation.
   - When selecting a face of the source, all the body containing the pad is added to the simulation.
3. Set the parameters (see page 105).
4. Click OK.
   - The interactive simulation appears in the specification tree.
   - The interactive simulation appears in the 3D view.
   - You can reframe on, hide, show, edit, copy, paste or delete (see page 123) the simulation.
   - You can edit the simulation properties (see page 124).

Note that you must click Local Update to update the simulation.

For more details, you can view Managing the Simulations Objects. (see page 123)
Note that when creating a new simulation, you can copy every compatible sources, geometries or sensors of an already created simulation. To do it, you must select the already created simulation from the specification tree or in the graphics area.

**Interactive simulation**

**Parameters of an Interactive Simulation**

- You can select the Preview meshing check box.
  
  For more details about preview meshing, you can view Setting the Simulation Properties (see page 138).
- From the Light expert list, you must select true or false.
  
  For more details about light expert, you can view Using Light Expert (see page 105).
- In the Ambient Material box, you must select an ambient material by clicking it in the graphics area or by selecting it in the specification tree.
  
  For more details about ambient material, you can view Ambient Material (see page 20).
- If you want to use simulation preset, you must click Select.
  
  For more details about simulation presets, you can view Presets (see page 146).
- If you click More>>, in the Visualization as geometry box, you can select visualizations of files.
  
  For more details, you can view Visualization as Geometry (see page 105).

**Using Light Expert**

With Light Expert, you can perform ray tracing filtering on an interactive simulation in order to analyze some particular light path.

**You must have created an Interactive, Surface or Ray File Source to use this parameter.**

1. In the Interactive Simulation dialog box, select true from the Light Expert list.
2. Click OK.
3. Click Local Update to update the simulation.

   The .lpf file appears in the specification tree.

For more details about LPF files, you can view Visualizing a LPF Result for Interactive Simulation (see page 160).

**Using Visualization as Geometry**

Geometries without specifications, such as VRML, STL, 3DXML, CGR or CATIA V4 model can be used directly in simulations.

**You must apply materials at the product level.**

**You must apply a material to all geometrical elements.**
If you select a material with optical properties, the optical properties are used.

If files contain graphical characteristics as texture, they are applied to the geometry.

Material with optical properties (CGR & VRML files)

You can select a SafeWorks Manikin to use its meshing as geometry. As the manikin does not have graphical material, the only way to use it within a simulation is to apply a CATIA material.

You must have created a manikin in a CATProduct.

1. Apply a material to the manikin.
   The material can contain optical or rendering properties.
   If textures are included in the simulation, they are taken into account.
2. Insert the CATProduct in the assembly of the simulation.
3. From the simulation definition dialog box, in the Visualization as geometry box, select the manikin from the specification tree.
4. Launch the simulation.

Export Simulation Rays as Geometry

With Export Simulation’s Rays as geometry, you can export rays into geometries.

The geometry set is composed of CATIA V5 polylines per ray with one point per impact

You must have created an interactive simulation first.

Exporting Simulation Rays as Geometry

1. Click Export Simulation’s Rays as Geometry (Simulations).
   Note that you can also right-click an interactive simulation in the specification tree, select Interactive Simulation Object, and then click Export Simulation’s Rays as Geometry.
2. Set the parameters (see page 106).
3. Click OK.
   The geometry set appears in the CATIA specification tree.
   The geometry set appears in the 3D view.

Parameters of Export Simulation Rays as Geometry

- In the Export Rays from group box, you must select an interactive simulation by clicking it in the specification tree or in the graphics area.
• In the Geometrical elements will be placed in a Part group box, you must create a new part or select an existing one.
• If you select the Create a new Part under Product check box, you must select a product from the specification tree.
  The geometry is inserted in a new part document created and inserted under that product.
• If you select the Select an existing Part check box, you must select a part from the specification tree.
  The geometry is added to that part document.

Direct Simulation
With Direct Simulation, you can propagate a large number of rays from sources to sensors and through an optical system.
At the end of a simulation, photometric or colorimetric levels are available for measurement.
With a Direct Simulation, you may face with OPTIS Distributed Computing Restrictions.

Creating a Direct Simulation
With a Direct Simulation, you can trace light rays from a source to a sensor.
1. Click Direct Simulation (Simulations).
2. Select Sources, Geometries and Sensors involved in this simulation.
3. Set the parameters (see page 107).
4. Click OK.
   The direct simulation appears in the specification tree.
   You can reframe on, hide, show, edit, copy, paste or delete (see page 123) the simulation.
   You can edit the simulation properties (see page 124).

Note that you must click Local Update to update the simulation.
For more details, you can view Managing the Simulations Objects (see page 123).
Note that when creating a new simulation, you can copy every compatible sources, geometries or sensors of an already created simulation.
To do it, you must select the already created simulation from the specification tree or in the graphics area.

Parameters of a Direct Simulation
• You can select the Preview meshing check box.
  With Preview Meshing, you can display the meshing of individual components.
  For more details about meshing, you can view Setting the Simulations Properties (see page 138).
• In the Number of rays box, you must type or edit a value.
  Number of rays contained in an appearing ray file can be adjusted.
  Ray number for direct simulation can be greater than 2 Giga Rays.
• From the Ray file list, you must select true or false.
  If you select true, you create a .ray file that appears in the specification tree.
• From the Light expert list, you must select true or false.
  
  For more details about light expert, you can view Using Light Expert (see page 108).

• In the Design Table box, you can select a design table for multi-configuration.
  
  For more details about design table, you can view Using a Design Table with a Simulation (see page 108).

• In the Ambient Material box, you must select an ambient material by clicking it in the graphics area or by selecting it in the specification tree.
  
  For more details about ambient material, you can view Ambient Material (see page 20).

• If you want to use simulation preset, you must click Select.
  
  For more details about simulation presets, you can view Presets (see page 146).

• If you click More>>, in the Visualization as geometry box, you can select visualizations of files.
  
  For more details, you can view Visualization as geometry (see page 108).

Using the Light Expert

With Light Expert, you can identify some particular light path reaching a sensor area.

1. In the Simulation Definition dialog box, select true from the Light expert list.

2. Set a value in the LXP max paths spinbox.

   LXP max paths is the number of rays which is displayed.

3. In the Sensors box, select a sensor and click LXP On/Off to turn on or off light expert for this sensor.

4. Click OK.

5. Click Local Update to update the simulation.

   The .lpf file appears in the specification tree.

   For more details about LPF files, you can view Visualizing a LPF Result for Direct or Inverse Simulation (see page 161).

Using a Design Table with a Simulation

With Design Table, you can launch different simulation on different configurations of an optical system.

1. In the Simulation Definition dialog box, you must select the design table configuring the optical system.

2. Click OK.

3. Click Local Update to update the simulation.

   Note that the current configuration is indicated in the progress dialog box.

   The results appear in the specification tree.

Using Visualization as Geometry

Geometries without specifications, such as VRML, STL, 3DXML, CGR or CATIA V4 model can be used directly in simulations.

You must apply materials at the product level.

You must apply a material to all geometrical elements.

If you select a material with optical properties, the optical properties are used.
If files contain graphical characteristics as texture, they are applied to the geometry.

*Material with optical properties (CGR & VRML files)*

You can select a SafeWorks Manikin to use its meshing as geometry. As the manikin does not have graphical material, the only way to use it within a simulation is to apply a CATIA material.

**You must have created a manikin in a CATProduct.**

1. Apply a material to the manikin. The material can contain optical or rendering properties. If textures are included in the simulation, they are taken into account.
2. Insert the CATProduct in the assembly of the simulation.
3. From the simulation definition dialog box, in the Visualization as geometry box, select the manikin from the specification tree.
4. Launch the simulation.

**Inverse Simulation**

With an Inverse Simulation, you can propagate a large number of rays from a camera or an eye (Radiance Sensor) to sources and through an optical system. At the end of a simulation, photometric or colorimetric level are available for measurement.

With an Inverse Simulation, you may face with OPTIS Distributed Computing Restrictions.

**Creating an Inverse Simulation**

With an Inverse Simulation, you can trace rays from a sensor to a source.

1. Click Inverse Simulation (Simulations).
2. Select Sources, Geometries and Sensors involved in this simulation. You can use only one single sensor for each simulation.
   - If you include a Sensor Camera, you do not have to select a source.
   - If you include a Geometric Sensor Camera in an inverse simulation, you cannot include another sensor type.
   - You can only select luminance, radiance or irradiance sensors. Irradiance sensors are only available for Monte Carlo inverse simulations.
3. Set the parameters (see page 110).
4. Click OK.
   - The inverse simulation appears in the specification tree.
The inverse simulation appears in the 3D view.
You can reframe on, hide, show, edit, copy, paste or delete (see page 123) the simulation.
You can edit the simulation properties (see page 124).

Note that you must click Local Update to update the simulation.
For more details, you can view Managing the Simulations Objects (see page 123).
Note that when creating a new simulation, you can copy every compatible sources, geometries or sensors of an already created simulation.
To do it, you must select the already created simulation from the specification tree or in the graphics area.

Parameters of an Inverse Simulation

- You can select the Preview meshing check box.
  For more details about preview meshing, you can view Setting the Simulation Properties (see page 138).
- In the Number of pass box, you must type or edit a value.
  Number of pass only appears in case the Monte-Carlo is true (see page 134).
  For details about the number of pass, you can view Virtual Photometric Lab.
- From the Light expert list, you must select true or false.
  For more details about light expert, you can view Using Light Expert (see page 110).
- In the Design Table box, you can select a design table for multi-configuration.
  For more details about design table, you can view Using a Design Table with a Simulation (see page 111).
- In the Ambient Material box, you must select an ambient material by clicking it in the graphics area or by selecting it in the specification tree.
  For more details about ambient material, you can view Ambient Material (see page 20).
- If you want to use simulation preset, you must click Select.
  For more details about simulation presets, you can view Presets (see page 146).
- If you click More>>, the Visualization as geometry box, the Out paths faces box and the In/Out button appear.
  - In the Visualization as geometry box, you can select the visualization of a file.
    For more details, you can view Visualization as geometry (see page 111).
  - In the Out paths faces box, you can select different transparent elements usable for outdoor light filtering by clicking them in the graphics area:

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

Directly click them in the graphics area, or use the Other Selection option (see page 11).
You can select the Mouse-over multi-selection mode check box to easily select a lot of faces.
You must click the first element to start the selection, move the pointer over every needed face, and then click the last element when the selection is complete.
If some elements must be removed from the selection, you must click to clear the Mouseover multi-selection mode check box and click unwanted faces to remove them.
For more details, you can view Using Outdoor Light Filtering (see page 112).
- The In/Out button is useful during a sun load study in a cockpit.
  With this parameter, you can define if the sources involved in the simulation are located inside or outside the geometries.
  You must click In/Out and the location of the source appears in brackets in the Sources box.

Using Light Expert

With Light Expert, you can identify some particular light path reaching a sensor area.
1. In the Simulation Definition dialog box, select true from the Light expert list.
2. Set a value in the LXP max paths spinbox.
   - LXP max paths is the number of rays which is displayed.
3. In the Sensors box, select a sensor and click LXP On/Off to turn on or off light expert for this sensor.
4. Click OK.
5. Click Local Update to update the simulation.
   - The .lpf file appears in the specification tree.

For more details about LPF files, you can view Visualizing a LPF Result for Direct or Inverse Simulation (see page 161).

**Using a Design Table with a Simulation**

With Design Table, you can launch different simulation on different configurations of an optical system.
1. In the Simulation Definition dialog box, you must select the design table configuring the optical system.
2. Click OK.
3. Click Local Update to update the simulation.
   - Note that the current configuration is indicated in the progress dialog box.
   - The results appear in the specification tree.

**Using Visualization as Geometry**

Geometries without specifications, such as VRML, STL, 3DXML, CGR or CATIA V4 model can be used directly in simulations.

*You must apply materials at the product level.*

*You must apply a material to all geometrical elements.*

If you select a material with optical properties, the optical properties are used.

If files contain graphical characteristics as texture, they are applied to the geometry.

**Material with optical properties (CGR & VRML files)**

You can select a SafeWorks Manikin to use its meshing as geometry.
As the manikin does not have graphical material, the only way to use it within a simulation is to apply a CATIA material.

*You must have created a manikin in a CATProduct.*

1. Apply a material to the manikin.
   - The material can contain optical or rendering properties.
   - If textures are included in the simulation, they are taken into account.
2. Insert the CATProduct in the assembly of the simulation.
3. From the simulation definition dialog box, in the Visualization as geometry box, select the manikin from the specification tree.
4. Launch the simulation.

**Outdoor Light Filtering**

Outdoor Light Filtering concerns sun load studies in cockpits. Initially, the parts composed by all the selected geometries blocked the light coming from the sun, the sky or other outdoor lights. With Outdoor Light Filtering, you can specify only the glazing parts avoiding the gathering and the selection of all opaque parts. With Outdoor Light Filtering, you can gain both preparation and simulation time.

1. In the Inverse Simulation Definition dialog box, click More >>.
2. In the Out paths faces box, click the graphics area to select faces.
3. Click OK.
4. Click Local Update to update the simulation.

The result appears in the specification tree.

---

**Virtual BSDF Bench**

With the Virtual BSDF Bench, you can obtain a BSDF measurement from a virtual texture in order to apply it to the 3D surface and get a first idea how it is going to look like using Visual Ergonomics. It consists in creating a new simulation that would allow selecting 3D surfaces as well as setting source and sensor’s parameters.

With the Virtual BSDF Bench, you cannot use OPTIS Distributed Computing.
Using the Virtual BSDF Bench

1. Click Virtual BSDF Bench (Simulations).
2. Set the parameters (see page 113).
3. Click OK.

The virtual BSDF bench simulation appears in the specification tree.

Note that you must click Local Update to update the simulation.
For more details, you can view Managing the Simulations Objects (see page 123).

Parameters of Virtual BSDF Bench

*The static distribution of surface heights has to be planed.*

The X and Y dimensions of the surface are much superior to the average width of roughness and there are a large number of roughnesses.

All angle values have to be given in degrees between 0 and 360 values.

- In the Design Table box, you can select a design table for multi-configuration.
  
  For more details about design table, you can view Using a Design Table with a Simulation (see page 118).
- If you want to use simulation preset, you must click Select.
  
  For more details about simulation presets, you can view Presets (see page 146).

Simulation

Format Description

In the Format description group box, you must select the file description that can be anisotropic BSDF, complete scattering or unpolished file.

Depending on the studied surface features, different surface models can be chosen in order to model the surface BSDF properties. The main principle is to trace rays from a discrete set of user defined incidences and statistically study the way they interact with the surface. The data is then collected and converted in one of our BSDF surface model.

- When selecting Color does not depend on viewing direction, after the BRDF calculation, a coefficient of spectral adjustment is applied and an anisotropic BSDF file is generated as .anisotropicbsdf file.
  
  This format also supports isotropy. You must select the Anisotropic check box only in case the surface you want to study requires anisotropy as stripes or grid for example.
  
  In case the anisotropy of the surface is not too pronounced, it is possible to neglect it by running an isotropic simulation: the software averages the behavior of the anisotropy over all surface orientations.
  
  This model is suited for surface with little or no dependence on wavelength that is the dependence on wavelength can be neglected as for some metals, white, grey or black paints or plastics...
  
  For more details about anisotropic BSDF format, you can view Anisotropic BSDF Surface.

- When selecting Color depends on viewing direction, a BRDF calculation is done for each wavelength and a complete scattering file is generated as .brdf file.
  
  This model can be used when the studied surface’s spectral behavior is not suited for the anisotropic BSDF model as for colored plastic with specular component, shiny colored paints...
  
  This model drops the possibility of an anisotropic surface and only supports isotropic surfaces.
  
  In case the anisotropy of the surface is not too pronounced, you can still use this option. The software averages the behavior of the anisotropy over all surface orientations.
  
  For more details about complete scattering format, you can view Complete Scattering Surface.
Just like for anisotropic BSDF, in case the surface reflects light differently on each side, clicking BSDF depends on light incidence side makes the simulation light the surface on both sides to study both behaviors.

As an example, one side is blue and when you return the surface, it is white.

In this case, the file extension also changes to .bsdf180.

- When selecting BSDF depends on surface roughness only, an unpolished file is generated as .unpolished file.

This mode of operation is in complete rupture with the two previous ones.

- Object has to be a transparent material with diffuse properties depending only on surface geometry.
- You must upload this geometry in your CAD model.
- There is no volume diffusion and the roughness is quite low.
- Optical properties of the material surface have been defined by Mirror with a one hundred percent of reflectance, for the transmitted rays not to interfere with the reflected distribution.

This mode is mainly aimed at studying rough transparent surfaces whose geometry was measured by profilometry.

For more details about unpolished surface, you can view Unpolished Surface.

In order not to have the transmitted rays to interfere with the reflected distribution you want to record, the optical properties must be set as opaque material and perfect mirror 100% surface quality.

For more details, you can view Adding Optical Properties To a Material.

This also makes sure no rays are lost in transmission. All rays are reflected making the simulation more efficient.

In the real world, this is equivalent to a thin metallic film deposition.

Afterwards, when the resulting .unpolished file is used in a photometric simulation, the Fresnel formulas are applied in conjunction with the normal distribution to rebuild both the reflected and transmitted ray distributions.
Reconstructed ray distribution

The great advantage of this approach is that the resulting .unpolished file is independent of the material it is applied to. At measurement time, the effect of the material index has been removed by the perfect mirror surface property. Then the effect of the refraction index is only reintroduced at simulation time. This can prove very useful in the case of molded plastics. One can use the same mold using PMMA or PC for example. In this case, the geometric shape of both is the same but their optical properties are different due to their respective refraction indexes. In a photometric simulation, both surfaces can be modeled using the same .unpolished file. The model adapts to the VOP material of the body it is applied to. One can even try other types of plastic to simulate their optical performances before they are actually molded. This approach has limitations though. It only works properly as long as there are not too much rays having more than one interaction with the surface.

Multiple interactions

This behavior appears when the surface roughness becomes high. So this model must not be used in case the diffuser has a lambertian like behavior. Only the gaussian specular behavior is properly modeled.

Wavelength

In Wavelength group box, you can type or edit the wavelength values.
Values have to be the center of the limit interval.

In Sampling box, you can type or edit the sampling value.

Number of Rays

You can set a value in the Number of rays spinbox.
It is recommended to choose at least 10 000 rays.
Number of rays is the number of rays per configuration.
In case of Color depends on viewing direction format description, value is the number of rays per incident angle and per wavelength.
Otherwise value is the number of rays per incident angle but for all wavelengths. As a consequence, for a constant number of rays, the simulation time varies belong to the other parameters: number of wavelength, number of theta and phi angles of the source, output file type, average number of rays interaction in the material and symmetry properties of the surface.

Geometry

1. In the Geometries list, you must select a geometry in the graphics area.
You can select the following elements:

| Faces | Surfaces | Solids |
Faces
directly click them in the graphics area, or use the Other Selection option (see page 11).

-Or-

2. In the Visualization as geometry box, you can select the visualization of files.
   -For more details, you can view Visualization as geometry (see page 118).
   -You can select the Preview meshing check box.
     -For more details about preview meshing, you can view Setting the Simulation Properties (see page 138).
   -In the Axis System group box, you must set the parameters.
     You can click Reverse direction to reverse the direction.
     -Axis must define a parallel plan to the average plane of the surface.
   -In the Analysis Area group box, you must type the X and Y Ratio values or change them by using the arrows.
     -These values are used to remove border effects.
     -X and Y Border ratio dimensions determines the surface on which measurements are going to be made and determines the size of light source that is calculated at each step according to the apparent surface of the sample.
     -X and Y Size dimensions are calculated using the size of the bounding box around the sample to measure and the X and Y border ratios.
     -If needed you can select the Identical ratios check box.

Source

- From Theta group box, in Sampling box, you can type or edit the sampling value.
  - Theta start value is fixed and equal to 0deg.
  - Theta end value can take two values according to the type of measurement: 90deg or 180deg.
- From Phi group box, in Sampling box, you can type or edit the sampling value and you must select the symmetry type.
  - Phi group box is available only in case of anisotropic format.
  - Phi start is fixed and equal to 0deg.
  - Phi end varies according to the scattering symmetry of the measured sample.

<table>
<thead>
<tr>
<th>PHI END</th>
<th>SYMMETRY</th>
</tr>
</thead>
<tbody>
<tr>
<td>360</td>
<td>No symmetry</td>
</tr>
<tr>
<td>180</td>
<td>Symmetry to plane 0-180</td>
</tr>
<tr>
<td>90</td>
<td>Symmetry to planes 0-180 and 90-270</td>
</tr>
</tbody>
</table>

- In Adaptive sampling group box, you can browse an adaptive sampling file containing incident angles not regularly distributed.
When using an adaptive sampling file, it has to be correctly formatted according to the format parameter.

- Line 1 is a header.
- Line 2 is a comment.
- Line 3 contains the number of Theta angle values.

### The minimal value is 3.

- Line 4 contains the Theta angles list in degree and starting with the 0 value.

  The Theta maximal value is usually 90 or 180 degrees to have the source lighting also the other face of the BSDF180 surface.

  In case of 90 value, the value is not taking into account in the calculation because it is not a physical value.

- Line 5 contains the number of Phi angle values.

### The minimal value is 2 in case of anisotropic format.

For other formats, value can be 0 or 1 and line 6 will not be read.

- Line 6 contains the Phi angle list in degree and starting with the 0 value. The maximal value is 360 degrees.

### File example in case of anisotropic format.

```
OPTIS - Incident sampling file v1.0
Comment
10
 0 2 4 6 8 10 20 40 60 80
12
 0 10 20 30 50 80 120 160 200 250 300 330 360
```

### Sensor

- In the Type box, you must select Reflection and Transmission or Reflection only.
  
  Reflection and Transmission corresponds to BSDF measurements and Reflection only to BRDF measurements.

  In case of unpolished type, you must select Reflection only.

- In the Integration angle box, you can type or edit the value in degrees.

  If needed, you can select the Automatic sampling check box to automatically calculate the sampling during the simulation.

  Size of result files is reduced keeping only required measurement points at the sensor level.

  The Start and End parameters of Theta and Phi would be used as well as Sampling but the samples would be chosen adaptively.

- From Theta group box, in Sampling box, you can type or edit the sampling value.

  Theta start value is fixed and equal to 0deg.

  Theta end value can take two values according to the type of measurement: 90deg or 180deg.

- From Phi group box, in Sampling box, you can type or edit the sampling value.

  Phi start value is fixed and equal to 0deg.

  Phi end value is fixed and equal to 360deg.

The sampling of the Theta and Phi angles have to be better than the integration angle otherwise some rays are not integrated.

- In Adaptive sampling group box, you can browse an adaptive sampling file.

  When using an adaptive sampling file, it has to be correctly formatted according to the format parameter.

  - Line 1 is a header.
  - Line 2 is a comment.
  - Line 3 contains the minimal Theta angle value.

### Value has to be 0.

- Line 4 contains the maximal Theta angle value.

  The Theta maximal value is 90 or 180 degrees.

- Line 5 contains the number of constant areas for the Theta sampling.
Line 6 contains for each area the limit angle value and the sampling step value.
In the following example, Theta angle varies from 0 to 86 degrees and is sampled belong to three areas: from 0 to 10 degrees every 0.5 degrees, from 10 to 30 degrees every degrees, and from 30 to 86 degrees all 2 degrees.

The maximal angle value has to be equal to the line 4 value.

Line 7 contains the number of constant areas for the Phi sampling.

Line 8 contains for each area the limit angle value and the sampling step value.
In the following example, as Phi angle varies from 0 to 360 degrees, there is no limits values and the Phi angle is sampled belong to one area every 5 degrees.

File example

```
OPTIS - Detector sampling file v1.0
Comment
0
86
3
10 0.5 30 1 86 2
1
360 5
```

Using a Design Table with a Simulation

With Design Table, you can launch different simulation on different configurations of an optical system.

1. In the Simulation Definition dialog box, you must select the design table configuring the optical system.
2. Click OK.
3. Click Local Update to update the simulation.
   Note that the current configuration is indicated in the progress dialog box.
   The results appear in the specification tree.

Using Visualization as Geometry

Geometries without specifications, such as VRML, STL, 3DXML, CGR or CATIA V4 model can be used directly in simulations.

* You must apply materials at the product level.
* You must apply a material to all geometrical elements.

If you select a material with optical properties, the optical properties are used.

If files contain graphical characteristics as texture, they are applied to the geometry.
Material with optical properties (CGR & VRML files)

You can select a SafeWorks Manikin to use its meshing as geometry. As the manikin does not have graphical material, the only way to use it within a simulation is to apply a CATIA material.

You must have created a manikin in a CATProduct.

1. Apply a material to the manikin.
   - The material can contain optical or rendering properties.
   - If textures are included in the simulation, they are taken into account.
2. Insert the CATProduct in the assembly of the simulation.
3. From the simulation definition dialog box, in the Visualization as geometry box, select the manikin from the specification tree.
4. Launch the simulation.

Interpolation Enhancement Overview

BSDF measurements are done at finite incidence angles determined by user.

For more information, you can view Defining Incidence Angles.

In simulation, the BSDF data file applied as optical surface property to a given surface contains data that allow SPEOS to know how a ray arriving with one of the incidence angles at the surface is reflected and diffused, however to manage a ray arriving at intermediate incidence angle SPEOS requires the BSDF data to be interpolated.

The following example illustrates the BSDF interpolation between 10° and 20° incidence angles. We can see here two specular peaks at 10° and 20° instead of one peak at 15°.

After interpolation the BSDF data for incidence angle 15° appears and is properly oriented.

Note that most of the diffuse part stay in place contrary to the specular peak that rotates according to the incidence angle.

In addition to the BSDF interpolation that is done with the BSDF viewer, you have the opportunity to help SPEOS distinguishing specular and diffuse components. Specular and diffuse components are managed with different algorithms to speed up simulations.
A white wireframe cone is used to differentiate the specular part (shown inside the cone) from the diffuse part (shown outside the cone).

**Interpolating BSDF measurements**

*You must have OMS2 or OMS4 software to use interpolation enhancement.*

The use of Interpolation Enhancement has to be performed only once per measure.

*You must open a measured file to use Interpolation Enhancement.*

1. Click File, Interpolation enhancement.
   
   A window appears.
2. Set the parameters (see page 120).
3. Click OK.
4. Save the file with a different name.
5. Close the file.
6. Open the file you just saved to see the interpolation on the BSDF data.

**Parameters of Interpolation Enhancement**

**View Control**

In View control box, you must select Full view, Specular part or Diffuse part to switch to a specular or a diffuse view of the BSDF. This helps to visualize the boundary between the specular and the diffuse parts.

The optimal cone position is probably achieved when the diffuse part looks like as much as possible a lambertian distribution.

**Specular Versus Wavelength**

You can also select Keep specular constant versus wavelength to get a constant specular reflection with an anisotropic BSDF model. Set the wavelength value in the Base wavelength box.

This option is not available in case of a complete scattering surface (BRDF).

For more information, you can view Specular Constant for Anisotropic BSDF.

**Cone Properties**

In Cone properties box, you must use Angle and Height of cone sliders to adapt the BSDF.

When the cones are saved, they can be used both in the viewer and during the simulations to achieve an optimal interpolation between the incidences.
Parameters
Number of cones depends on whether there is transmission or not, number of incidence angles and number of wavelength or anisotropic angles.

In Parameters box:
- By clicking Reflection, you can change the mode between reflection and transmission.
- By changing Incidence values, you can change the incidence sample.
- By changing Wavelength values, you can change the wavelength sample (or anisotropic sample).

Examples

View Control: Full view
Cone properties: Angle: 25°; Height: 47

The height of the cone seems to be correct. It cuts the BSDF at the base of the specular peak.
The angle of the cone is too wide compared to the width of the specular peak. This may reduce the calculation speed.

After reducing the angle down to about 6° we can see the following.

View Control: Full view
Cone properties: Angle: 6°; Height: 47

We can check specular and diffuse parts look both alright.
If it is not possible to see the cone, it is likely the angle or height of the cone is too small. You must increase both until you can see the cone.

When the height is too small, you can see a crater in the diffuse part. You must increase the height until the crater disappears.

View Control: Diffuse part

When the height is too big, you can see a bump on the diffuse part. You must decrease the height until the bump disappears.

View Control: Diffuse part

When the angle is too small in the Full view, the cone totally disappears in the BSDF shape. You must increase the angle to make it only slightly larger than the specular peak.

View Control: Full view

It happen the angle is too large. The surface of the base of the cone is large compared to its intersection with the specular peak. This situation does not provide bad results but you must reduce angle to have faster performances.

View Control: Full view
Software performs faster if the base of the cone is only slightly larger than the specular peak.

**Simulations Management**

**Managing the Simulations within the Specification Tree**

All the created simulations appear in the specification tree. You can edit simulation definition by double-clicking on the simulation. You can right-click on the simulation to manage the simulation.

- If you want to focus on a simulation in the specification tree, click Center graph.
- If you want to focus on a simulation in the graphic area, click Reframe On.
- If you want to hide or display the simulation, click Hide/Show.
- If you want to edit the simulation properties, click Properties.
  
  For more details, you can view Simulations Properties (see page 124).
- If you want to cut, copy or paste the simulation, click Cut, Copy, Paste or Paste Special....
- If you want to delete the simulation, click Delete.

**Managing the Simulations Sub-Tree**

To open a simulation sub-tree from the specification tree, you can double-click on a simulation, or right-click on a simulation and click Open Sub-Tree.

- From simulation sub-tree, you can double-click on simulation parameters to type a value or change it by using the arrows.
- From Rays File and LXP, you can select true or false.

Note that all the simulations' results and reports appear in the specification tree.

**Managing the Simulations Objects**

By right-clicking on a simulation from the specification tree, you can click Simulation Name object.

When you rename a simulation, all the associated results are accordingly renamed at the same time.

- If you want to edit the simulation definition, click Definition....
- If you want to gather several simulation objects into a same sub-group, click Create Set (see page 11).
- If you want to run or update a simulation, click Local Update (Update).

A panel indicates the progress of the simulation.

Error XX% corresponds to the evolution of the total error number during the simulation for direct or inverse simulations.

You can click Cancel to stop the Simulation. Intermediate results with the current progress of the simulation are available.

You can expand the simulation node to access to the results.

- If you want to update a series of simulations without blocking the process if one of them leads to an error, click Local Update Without Notification.
• If you want to activate or deactivate a component, click Activate / Deactivate Component.
• If you want to move up and down the simulation in the Simulation section of the specification tree, click Move up or Move down.
• If you want to isolate, export or isolate and export a direct or an inverse simulation, respectively click Isolate, Export or Isolate and Export.
• If you want to export rays into geometries from an interactive simulation, click Export Simulation's Rays as Geometry.
• If you want to directly open the folder where an isolated simulation is located, click Open folder.

Simulation Properties
To edit the simulations properties, you can:
1. Select a simulation in the specification tree or in the 3D view.
2. Click Edit SPEOS properties.
   The properties of the simulation open.
-Or-
1. Right-click a simulation in the specification tree.
2. Click Properties.
3. Click More >>.
4. Edit the SPEOS properties of the simulation in the corresponding tabs.

Setting the Interactive Simulation Properties
Interactive Simulation tab is available for Interactive Simulation.
Interactive Simulation has been enhanced with impacts visualization. Only impacts on surfaces are displayed.
You must click More... to edit the Interactive Simulation tab.

• From the Draw rays list, you must select true or false.
• From the Draw impacts list, you must select true or false.

Interactive simulation with draws impacts

Interactive simulation with draw rays and draw impacts

Setting the Direct Simulation Properties

You must click More... to edit the Direct Simulation tab.

• From the Gathering radiance sensor list, you must select true or false.

With this parameter, you can improve the simulation performances. It enables the sensor gathering, which means that it aims at improving the convergence rate for radiance or luminance sensors in the case of direct simulations.

• If you select false from the list, the sensors are waiting for incoming photons to hit them.

This configuration may have a low probability depending on your system.

• If you select true from the list, for each photon propagating in the system and each impact on a surface, an additional ray is cast from the impact towards the sensor's observer point increasing the probability to have a signal in the map.

Generally it is better to set this parameter to true when the Radiance Sensor sees diffusing surface and to set to false in the other case.

When sensor gathering is active, only specular rays suffer from the integration angle approximation. The effect is a blurring of the specular signal.

• In the Save intermediate maps every box, you must type a value or change it by using the arrows.

With this parameter, you can save the intermediate results of a direct simulation.
Setting the Inverse Simulation Properties

Setting the Determinist Calculation Properties for an Inverse Simulation

You must click More... to edit the Inverse Simulation tab.

You must deactivate the Monte Carlo algorithm.

Determinist calculation does not manages dispersion or bulk diffusion and does not support light expert analysis.

Using Determinist without Photon Map

You must activate No Photon Map.

A simulation without photon map avoids any noise and does not manage the diffuse inter-reflection.

Authorize the Use of Rendering Properties as Optical Properties

You can activate the use of rendering properties as optical properties.

For more details, you can view Using Rendering Properties as Optical Properties (see page 34).

Ambient Sampling

The parameter defines the sampling.

The sampling is the quality of the ambient source.

The greater this value is the better the quality of the result is but longer is the simulation.

The following table gives some ideas of the balance between quality and time.

A default value could be 20 and a value for good results could be 100.
Specular Maximum Impact Number
This number defines the maximum number of specular interaction.

Anti-Aliasing
You can activate the anti-aliasing, to reduce artifacts as jagged profiles and fine details but increasing the simulation time.

Specular Approximation Angle
For rendering purposes, it can be interesting to replace perfectly specular surfaces with a Gaussian. This gives better and faster results.

For more details, you can view Specular Approximation Angle.
The typical application is the rendering of automotive tail lamps lit appearance. For this application, a typical value would be 5 to 10 degrees.
Using Determinist with Build Photon Map

You must activate Build Photon Map.

It is safe to use Photon Maps when surfaces are Lambertian, very diffuse or specular.
It is unsafe to use Photon Maps when surfaces are Gaussian with a small FWHM angle.

A simulation with build photon map generates map noises and manages the diffuse inter-reflections.

Simulation with build photon map

The Photon Mapping is a luminance algorithm used to take into account multiples diffuse inter-reflections but in this case it is a two pass algorithm.

- The first pass is a Monte Carlo direct simulation. Indeed, the first step of this technique is to send photons from sources into the scene and to store them in a map (the Photon Map) during the propagation phase.
- The second pass is a deterministic inverse simulation and is called the Gathering phase. The photon map from the first pass is used to compute local radiance.

Simulation Results

As the first pass corresponds to a Monte Carlo direct simulation, photons are randomly drawn.
So, photon deposition on scene parts are different from one simulation to another, implying different photometric results in localized measurements as shown below.
For example, considering a 10x10 mm² ellipsoid measurement area, and running 20 simulations, a 2.3 cd/m² standard deviation is obtained on this measurement series.

**Simulation results**

![Simulation results](image)

### Propagation Parameters Specific to Build Photon Map

For more details about other parameters, you can view Using Determinist without Photon Map (see page 126).

When you compute the Photon Map, you can set parameters for each phase some photon’s propagation.

- During the first pass, you can choose the number of rays to send from the source to the scene.
  
  You must be careful that it is not the number of photon stored in the photon map thus. There is one photon per impact and one ray can have many impacts.

- You can also set the depth of photon propagation. These parameters pilot the photon density in the photon map.
  
  The propagation stops after the given number of surface interactions.

- Direct Photon Number represents the number of rays sent in the direct phase.
  
  You can type or edit the value.
- Direct Max Impact represents the max impact of rays in the direct phase. You can type or edit the value.

- During the second pass, photon map can be used to compute local radiance/luminance at a given position. In order to achieve this, photon map is asked to give more or less photons, indicated in the neighbors field. For a first try, the best is to use the two next parameters keeping Use Final Gathering and Final gathering Max Neighbors options to the false selection.

- Max Neighbors represents the number of photons from the photon map taken into account to calculate the luminance. You can type or edit the value.
- Max Search Radius represents the maximum distance from the luminance calculation's point to search for neighbors contribution.
  You can type or edit the value.

The Max Search Radius parameter could have a significant impact on the results according to the Max neighbors parameter setting.
For example of a wall with one face illuminated and the other face not illuminated and no transmitting any light.
In case of a sensor observing the no transmitting face, if the Max search radius is higher than the depth of the wall, the sensor gives some luminance values corresponding to the illuminated side of the wall.
In the following example, the effect of a too large max search radius in simulation results is described.

Note the white dots on the right illustration resulting from the relationship between max search radius and max neighbors.
For a given max neighbors value, if the max search radius is too small, the detector do not collect all the neighbors and gives noisy results, and vice versa if max search radius is fixed and the max neighbors value is too high.

\[
\begin{array}{c}
\text{Max search radius } = 10 \\
\text{Max search radius } = 100
\end{array}
\]

The following example is a more obvious one, with the default value of the max neighbors parameter and a varying max search radius.
If this last parameter is too small, the result becomes more noisy.

Note that in this example, the simulation time is going to increase as a function of the max search radius until a value of this parameter for which all the neighbors are found. The simulation time stays constant after this crucial step.

**More Propagation Parameters Specific to Build Photon Map**

You can use Use Final Gathering and Final Gathering Max Neighbors options only if you are confident with the two last parameters.

You must select True from the Use Final Gathering list to use these options.

- You can activate the Use Final Gathering for better final results.
  This is to use the photon map after one diffuse interaction. This parameter uses secondary rays.
Note that in this case, during the first diffusing impact, a splitting is used. Max Search Radius applies here too.

Final gathering comes with two additional parameters: final gathering max neighbors and splitting number.

The algorithm is most often nicer but always much slower. Diffuse transmission is not taken into account.

- In the Final Gathering Max Neighbors box, you can type or edit a value to pilot the number of neighbors after the secondary rays. They are used to compute the luminance for each splitted ray. The value usually used is 10.
- In the Splitting number box, you can type or edit a value to set the number of splitted rays. The value usually used is 15.

Note that if there is an ambient source, the splitting number is not taking into account, replaced by the ambient sampling value.

- You can activate the Fast Transmission Caustics to accelerate the simulation. This is used when in the system, transparent objects of a scene are flat enough to neglect the refraction effect on the direction of a ray (Windows, Windshield, ...).

The result is right only for flat glass (parallel faces). The convergence result is faster using the option. The effect of the refraction on the direction is not taken into account.

<table>
<thead>
<tr>
<th>WITHOUT FAST TRANSMISSION CAUSTICS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIRECT PHOTON NUMBER = 10000</td>
</tr>
<tr>
<td>SIMULATION TIME = 160</td>
</tr>
</tbody>
</table>

Here the number of photon is not greater enough to see the direct lighting of the floor by the sun.

<table>
<thead>
<tr>
<th>WITH FAST TRANSMISSION CAUSTICS</th>
</tr>
</thead>
<tbody>
<tr>
<td>DIRECT PHOTON NUMBER = 10000</td>
</tr>
<tr>
<td>SIMULATION TIME = 100</td>
</tr>
</tbody>
</table>

By activating the Fast Transmission Caustics option, the direct lighting of the floor by the sun appears.
WITHOUT FAST TRANSMISSION CAUSTICS
DIRECT PHOTON NUMBER = 1000000
SIMULATION TIME = 260

Here the number of photon is greater enough to see the direct lighting of the floor by the sun.

Setting the Monte-Carlo Calculation Properties for an Inverse Simulation
You must click More… to edit the Inverse Simulation tab.
You must activate the Monte-Carlo algorithm.

Monte Carlo:
- Can generate noisy results,
- Manages dispersion,
- Manages bulk diffusion,
- Manages multiple diffuse inter-reflections,
- Supports light expert analysis.

Using the Monte-Carlo Algorithm
Authorize the Use of Rendering Properties as Optical Properties
You can activate the use of rendering properties as optical properties.
For more details, you can view Using Rendering Properties as Optical Properties (see page 34).

Automatic Save Frequency
You can type or edit a value.
An inverse simulation can save intermediate results during the simulation. This functionality is useful to check intermediate results for long simulations. A save operation occurs after automatic save frequency passes of the simulation.
0 means that the result is saved only at the end of the simulation. Of course reducing the number of save operations increases the simulation performance.
In the case of very high sensor sampling the save operation can take the half of the simulation time when automatic save frequency is 1.
If this parameter value is 0 and if the simulation is stopped without finishing the current pass, no result is available.

**Gathering Source Number**
You can type or edit a value.

In inverse simulations, each ray is propagated from the observer point through the map and follows a random path through the system.

There is often a very small probability for a ray to hit a light source on its own.

To increase this probability, new rays are generated at each impact on diffuse surfaces. These rays are called shadow rays, they are targeted to each light source in the system and the program checks whether a direct hit on the sources is possible or not.

If nothing happens, the program finds a hit, it computes the corresponding radiance to store in the map.

The gathering source number parameter pilots the number of shadow rays to target at each source.

**Dispersion**
You can activate the dispersion.

With this parameter, you can activate the dispersion calculation. In optical systems in which the dispersion phenomena can be neglected, the colorimetric noise is canceled by deactivating this parameter.

Simulation involving materials modeled with dispersion cannot be done.

**Splitting**
You can activate the splitting.

If you do so, the path of each reverse propagated ray is split in more paths at the first impact after leaving the observer point. Further impacts along the split paths do not provide further path splitting. This feature is primarily intended to provide a faster noise reduction on scenes with optical polished surface as the first surface state seen from the observer. An observer watching a car rear lamp is a typical example of such a scene.

The split is only done to the first impact: two rays are split on optical polished surface. On other surfaces there may be more or less split rays depending on the surface model. The 2 rays are weighted using Fresnel's law.

We are considering either the transmitted or the reflected ray (only one of them, we pick one each time). The choice (R or T) is achieved using Monte Carlo: the probability for reflection is the Fresnel coefficient for reflection. So depending on the generated random number, the ray will be either reflected or transmitted...

Close to the normal incidence, the reflection probability is around 4%, which is low. This low probability makes that when we want to see the reflection of the environment, we observe a lot of noise. The splitting algorithm removes this noise by computing the first interaction without using Monte Carlo.

Only use this option for tail lamps.

**Maximum Gathering Error**
You can type or edit a value.

With this parameter, you reduce the simulation time for scenes with large number of sources and where each source contributes to illuminate a small area of the scene.

This value defines the level below which a source can be neglected.

For instance a value of 10 as means that all sources contributing less than 10% to the illumination of all sources is not taken in consideration.

0, the default value means that no approximation will be done.
You must take some precautions by using layer operations tool of the Virtual Photometric Lab. For instance if maximum gathering error is defined at 1% for a simulation and if the flux of a source is increased 10 times with the layer operations tool this means that maximum gathering error is now 10% for this source.

**Fast Transmission Caustics**

When in the system, transparent objects of a scene are flat enough to neglect the refraction effect on the direction of a ray (Windows, Windshield, ...), you can activate the Fast Transmission Caustics list to accelerate the simulation.

- The result is right only for flat glass (parallel faces).
- The convergence result is faster using the option.
- The effect of the refraction on the direction is not taken into account.

---

**Using Dispersion**

The dispersion parameter influences Monte Carlo algorithm in terms of ray number generated by pass and type of noise on results.

- If you do activate it, a number of rays, corresponding to the luminance sensor wavelength sampling, is propagated over the entire spectrum per pass.
• If you do not activate it, one ray is propagated over the entire spectrum defined in the luminance sensor settings per pass.

\[ \lambda_{\text{min}} \quad \lambda_{\text{max}} \]

The refractive index variation with wavelength is not taken into account when dispersion is set disabled. The algorithm takes the sensor's median wavelength to calculate the refractive index:

\[ \frac{\lambda_{\text{max}} + \lambda_{\text{min}}}{2} \]

However, this approximation is valid for most of non imaging systems. Both algorithms generate colorimetric and photometric noises. Comparing them both, we observe that:

• If you do activate this dispersion, noise has a nature more colorimetric.
• If you do not activate dispersion, noise has a nature more photometric.

In terms of rendering, noise obtained without dispersion is visually more appealing than the one with dispersion. With the dispersion parameter, a light source illuminates a prism so a luminance sensor could look at the refracted light.

<table>
<thead>
<tr>
<th>Dispersion</th>
<th>Interactive Simulation Result</th>
<th>Noise Type</th>
<th>Simulation Time (s)</th>
</tr>
</thead>
</table>

*Inverse simulation with dispersion*
<table>
<thead>
<tr>
<th>Dispersion</th>
<th>Interactive Simulation Result</th>
<th>Noise Type</th>
<th>Simulation Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>True</td>
<td><img src="image1" alt="Dispersion True" /></td>
<td>False</td>
<td>290</td>
</tr>
<tr>
<td>False</td>
<td><img src="image2" alt="Dispersion False" /></td>
<td>False</td>
<td>10</td>
</tr>
</tbody>
</table>

*Difference between noise types and simulation times due to each algorithm.*

**Setting the Simulation Properties**

Simulation tab is available for all the simulations.

*You must click More... to edit the Simulation tab.*
**VOP on Surface**

From the VOP on surface list, you must select true or false.

Note that in a clean modeling, a volume optical properties (VOP) should not be applied to a surface geometry. However, in some cases it could be difficult to create a solid from a set of faces, especially with imported geometry. With SPEOS CAA V5 Based software, you can use a set of faces joined in surface feature as a solid. For more details, you can view VOP on Surface Tutorial.

![Simulation with VOP on surface](image)

**Meshing and Tessellation**

Note that you can also access the Tessellation parameters from the menu, by clicking Tools, Options..., and then selecting a package from the tree to edit the Meshing tab.

With Tessellation settings, you can lighten memory resources and accelerate simulation in specific cases. These parameters define the default parameters for simulations and control display of 3D sensors.

- By selecting the Temporary show in visualization check box, you can activate the simulation meshing to display all the meshed components.

![Simulation meshing](image)

- From the Tessellation sag mode list, you must select Proportional or Fixed to define the meaning of the tessellation sag value.
- You must set a value in the Tessellation sag value spinbox. The sag value defines the maximum distance between a bar and the object to tessellate.
• If you selected Proportional from the Tessellation sag mode list, the sag value depends on the size of each face.
  
  Sag value = bounding box diagonal / sag ratio.

• If you selected Fixed from the Tessellation sag mode list, the sag value is equal to the tessellation sag mode value.

The sag value defines the maximum distance between the geometry and the tessellation.

![Sag 1 > Sag 2](image)

Tessellation sag value with proportional and fixed modes

• From the Tessellation step mode list, you must select Proportional or Fixed to define the meaning of the tessellation step value.

• You must set a value in the Tessellation step value spinbox.
  
  The step value defines the maximum length of a bar.

  • If you selected Proportional from the Tessellation sag mode list, the sag value depends on the size of each face.
  
  Step value = bounding box diagonal / step ratio.

• If you selected Fixed from the Tessellation step mode list, the step value is equal to the tessellation step mode value.

The step value defines the maximum size of a triangle of the tessellation.

![Step 1 > Step 2](image)

Tessellation step value with proportional and fixed modes

• You must set a value in the Tessellation angle spinbox.
  
  The angle value defines the maximum angle between the normal at each bar end.

![Angle 1 > Angle 2](image)

Tessellation angles

**Geometrical Parameters**

• In the Geometrical angle tolerance box, you can type or edit an angle value.
  
  This parameter defines the maximum angle to consider two faces as tangent.

• In the Geometrical distance tolerance box, you can type or edit a length value.
  
  This parameter defines the maximum distance to consider two faces as tangent.
**Maximum Impact Number**

- In the Maximum impact number box, you must type or edit a value.
  
  This parameter defines the maximum number of a ray's impacts to continue to propagate it. It is useful to stop the propagation of ray in optical system as an integrated sphere in which a ray is never stopped.

**Smart Engine**

In the Smart Engine box, you must type a value or change it by using the arrows.

- This parameter controls a Smart Engine parameter that defines a balance between the speed and memory.
  
  - For interactive simulation, the best value is 3 if there is no camera sensor, otherwise, 7 is recommended.
  
  - For direct simulation and inverse simulation, the best value is 9.

  It is not recommended to change this value for a classical use of SPEOS CAA V5 Based.

  However, in some cases when memory use is critical due to huge geometries (i.e., complete cockpit, cabin, car or building), this value can be reduced in order to save memory.

  Also, in other cases when a simulation contains very small detailed geometries inserted in a big scene (i.e., detail of headlamp bulb placed in a simulation with a 50m long road geometry), this value can be increased to reach better performances.

  It becomes interesting to use the smart engine parameter in case of a large number of rays. As an example, it is not the case for a Light Modeling interactive simulation with around 100 rays, and it is the case for a Digital Vision and Surveillance interactive simulation with around 300k rays.

**Disable Tangent Bodies Management**

From the Disable tangent bodies management list, you must select true or false.

- Note that SPEOS CAA V5 Based software is able to manage tangent solid.
  
  This management requires extra processing time.

  If an optical system does not have tangent faces, the simulation can be speeded up by setting this parameter to true.

  If disable tangent bodies management parameter is set to true and some faces are tangents, propagation errors are generated and results are wrong.

**Weight**

With Weight, you can efficiently control the energy of each ray.

- It is highly recommended to set this parameter to true excepted in interactive simulation.

  Unsetting this parameter is useful to understand some phenomena as absorption.

  You can activate the Weight parameter.

  If you do so, you must type a or edit value in the Minimum energy percentage box.

  The Minimum energy percentage parameter defines the minimum energy ratio to continue to propagate a ray with weight.

  This parameter helps the solver to better converge according to the simulated lighting system.

  This parameter influences Monte Carlo processing of Ray/Face and Ray/Volume interactions.

  If you do not activate weight, rays' energy stays constant and probability laws dictate if they continue or stop propagation.

  If you do activate weight, ray's energy evolves with interactions until they reach sensors.

  - In the case of Ray/Face interaction, you must consider rays reaching an optical surface having a 50% reflectivity.

    - If you do not activate weight, rays have 50% probability to be reflected.
If you do activate weight, all the rays are reflected with 50% of their initial energy.

- In the case of Ray/Volume interaction, you must consider rays propagating inside an absorbing material.
  If you do not activate weight, rays have a probability to be absorbed or transmitted according to their path through the material.
  If you activate weight, rays' energy decreases exponentially according to the material absorption value and the path of rays through it.

Practically, using weight in simulation improves results' precision as more rays with contributing energy reach sensors.
So, to get the same amount of rays on sensors without the Weight parameter, you need to set more rays in simulations, which also increases simulation time.

Direct simulation result when weight is true
Direct simulation result when weight is false

Not activating weight is very useful in two cases.
1. When you analyze phenomena such as absorption.
   Considering a material with absorption, here is the observation of the absorbed rays using an interactive simulation.
2. If you want a simulation performance improvement in closed systems, let us consider an integrating sphere with inside a light source and a sensor. The surface inside the sphere has a high reflectivity value. The system is set so the sensor is protected from direct illumination from the light source.

Note that the simulation duration depends on the true or false parameter. When weight is activated, simulation time corresponds to 1747. When weight is not activated, simulation time corresponds to 440.

This difference is due to the fact that low energy rays are still propagating after several bounds in the system for simulations using weight whereas the probability the rays still propagate decreases each bound they make for simulations not using weight.

Texture Normalization

With Texture, you can switch between different normalization methods and activate the taking into account of the textures applied on the geometry through materials in the SPEOS CAA V5 Based simulations. From the Texture list, you must select true or false. If you selected true from the list, you must select a normalization from the Texture normalization list.
Note that textures modify the interaction between light and faces for all kinds of interaction excepted the specular reflection.

- If you select false from the Texture list, only surface optical properties are taken into account in the simulation result.

![Simulation result when texture is false](image1)

- If you select true from the Texture list and None from the Texture normalization list, the simulation result takes into account BRDF or scattering properties of the simulation elements plus texture information.

![Simulation result when texture is true and texture normalization is none](image2)
• If you select true from the Texture list and Color from BRDF from the Texture normalization list, the simulation result is normalized using the BRDF or scattering properties and shows the texture on the elements. In average, the absorption from texture is null. With this option, you can get photometric results close to the ones obtained without texture.

Simulation result when texture is true and texture normalization is color from BRDF

• If you select True from the Texture list and Color from Texture from the Texture normalization list, the simulation result is normalized using the texture and shows it on the elements. The absorption from BRDF or scattering properties is null.

Simulation result when texture is true and texture normalization is color from texture

When looking globally, the texture does not have any effects on the photometry, but some pixels can have a superior or inferior value compared to the physical value.

Bump

You must have set bump properties in the CATIA rendering properties of the materials.
You must first set Texture to True.

Bump is recommended for visualization purposes and not for purely realistic physical renderings or photometric analysis.

With Bump, you can improve the rendering of your simulation results and get precise details, as the reliefs of a dashboard for example.
In CATIA, when defining the textures applied on a geometry, you can select the Image texture type and set a Bump
parameter to a relative height between -1 and 1, 0 corresponding to no relief. SPEOS CAA V5 Based can take this parameter into account during the simulation, to get enhanced visualization results.

The bumps defined within the Car Paint and External Shader texture types cannot be taken into account during simulations.

From the Bump list, you must select True or False to activate or deactivate the bump.

Simulation result without bump.

Simulation result with bump.

Presets

Preset tab is available for all simulations.

Note that when a preset is applied to a simulation, all the simulation parameters are locked.

You must click More... from the Properties dialog box, or click Select from a simulation definition dialog box to edit the Preset tab.

Understanding the Presets

Simulation Preset, predefined sets of parameters, are models of simulation settings.

You can create a preset from the parameters of an existing simulation, and you can apply it to another simulation. When creating preset, all the parameters common to any simulation types and specific to a certain type are copied. Simulation type and SPEOS CAA V5 Based version are then integrated in the preset in order to restrain its subsequent use as direct simulation presets for direct simulations.

A preset is then applied by copying each of its parameters in the corresponding simulation's parameter.

Presets are stored as .OPTPreset files in SimulationPresets directories.
All the presets are contained in a database accessible from the Simulation Presets tab (see page 148). Two types of library are managed:

- **Global**: the presets are common to user groups and cannot be modified (they can be set, for example, by advanced users for novices).
  
  If using Windows XP operation system, global presets are located in `C:\Documents and Settings\All Users\Application Data\OPTIS\SimulationPresets`.
  
  If using Windows Vista or Windows Seven operation systems, global presets are located in `C:\ProgramData\OPTIS\SimulationPresets`.
  
  If using Code Server installation system, global presets are located in `\Global\OPTIS\SimulationPresets`.

- **User**: the presets are only available for a local user and can be modified.

  If using Windows XP operation system, user presets are located in `C:\Documents and Settings\user\Application Data\OPTIS\SimulationPresets`.
  
  If using Windows Vista or Windows Seven operation systems, user presets are located in `C:\Users\user\AppData\Roaming\OPTIS\SimulationPresets`.
  
  If using Code Server installation system, user presets are located in `\User\OPTIS\SimulationPresets`.

**Preset**

- With the Preset interface, you can:
  
  - Visualize the preset applied to a simulation and the link status.
  - Save simulation parameters to make a new preset or modifying an existing one.
  - Select a preset to apply to the simulation.
  - Update an applied preset.
  - Suppress a link to a preset.

- If no preset is applied to the current object, you have two possibilities:
  
  - You can select Save parameters as Preset. You can type the preset name.
  - You can select Select an existing Preset. Then, you must select a preset from the list.

To validate an action, you must click Apply.

- If a preset is applied to a simulation, the preset's name and the link status are displayed in the Current Preset group box.

<table>
<thead>
<tr>
<th>LINK STATUS</th>
<th>MEANING</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ok</td>
<td>The preset is found and the simulation parameters are up to date.</td>
</tr>
<tr>
<td>To Update</td>
<td>The simulation parameters need to be synchronized to the preset.</td>
</tr>
<tr>
<td>Not Found</td>
<td>The preset is not found.</td>
</tr>
<tr>
<td>Version Non Compatible</td>
<td>The applied preset is not compatible with the used version of SPEOS CAA V5 Based.</td>
</tr>
</tbody>
</table>

- In the Actions group box, you can:
  
  - Select another preset of the list to apply.
  - Remove the link between the simulation and the preset.
  - Update the simulation when its status is not up to date.

Note that you can select several simulations to apply a preset.
Managing the Presets

With the Simulation Presets tab, you can manage the preset database. From the menu, you must click Tools, Options..., and then select a package from the tree to edit the Simulation Presets tab.

In the Preset Management section, you find two lists.

- The first list displays all the presets of the database, ordered by library type, and then by alphabetical order. You can select a preset in the list, activate the available commands and view the details of the selected preset.
  - With Copy Preset, you can copy a preset to a new one. All copied presets are created in the user library.
  - With Rename, you can rename an existing preset. Renaming a preset will automatically set all the simulations using it to not be up to date.
  - With Delete, you can suppress an existing preset.
  
  Note that copying, deleting and renaming a preset is only possible for user library type presets.

- With the the second list, you have the possibility to filter the displayed presets by library type and by preset type (direct, inverse,...). Note that in the Details section, you can view the type of preset and the SPEOS CAA V5 Based version used to generate it.

You can edit the preset settings. To do it, you must click Edit Settings... and then set the general simulation parameters and the specific type of simulation parameters. This automatically set all the simulations using the modified preset to not be up to date.

In the Default Presets section, you can define a default preset for each simulation type that is applied during the creating of a new simulation. For each simulation, you find:

- The name of the preset applied by default (<None> if there is no selected preset).
- A button to select or change the default preset.
- The information about the preset link status.

To change the default preset, you must click , and then select a preset in the list. You can also click the None button to remove the default preset.

If a default preset is not found, the Not found status and the GUID of the missing preset are displayed.

No preset with this status can be applied to the new simulations.

Understanding Propagation Errors

The goal of this task is to explain the different types of propagation error.

**Volume body not closed error**

Volume body not closed error occurs when a ray inside a solid body cannot hit a face of this given solid (to go out of this solid), only rays located very close to a corner can be involved. This error can occur either if the solid faces are not really closed (imported geometry) or if a ray cannot detect a face due to the geometrical optical precision.

This error can occur especially when VOP on Surface parameter is used. This error can be reduced by improving the precision of the CAD geometry or reducing the geometrical optical precision.

**Volume conflict error**

Volume conflict error occurs when a photon inside a body hits another solid body. This error can occur especially when VOP on Surface parameter is used.

This propagation error should be corrected by changing the geometry modeling.
2D tangent to 3D error
2D tangent to 3D error occurs when a solid geometry and surface geometry are tangent in a same simulation.

Note that SPEOS can manage two tangent solid geometries in a same simulation.
This propagation error can be avoided by separating solid and surface elements by at least geometrical optical precision.

2D intersect 3D warning
2D intersect 3D warning occurs when a surface body is crossing a solid.
This propagation error should be corrected by changing the geometry modeling.

Non optical material error
Non optic material error occurs when a ray enters in a solid body having non optic as volume optical properties.
This propagation error should be corrected by changing the modeling.

Non optical material at emission error
Non optical material at emission error occurs when a ray is directly emitted inside a solid body having non optic as volume optical properties.

Using Turbidity for an Ambient Source
This parameter is available in Ambient Source with Natural Light type.

Definition of turbidity (A Practical Analytic Model for Daylight, A. J. Preetham, Peter Shirley, Brian Smits):
Turbidity is a measure of the fraction of scattering due to haze as opposed to molecules.
This is a convenient quantity because it can be estimated based on visibility of distant objects.
We take the following example used in the tutorial ambient source. We consider the average luminance value of the measurement point indicated on the left plastic ball and we let the turbidity vary between 1.9 and 9.9.
<table>
<thead>
<tr>
<th>Turbidity</th>
<th>Normalized Simulation Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.9</td>
<td></td>
</tr>
<tr>
<td>2.5</td>
<td></td>
</tr>
<tr>
<td>3.1</td>
<td></td>
</tr>
<tr>
<td>Turbidity</td>
<td>Normalized Simulation Result</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td>3.7</td>
<td><img src="image1.png" alt="Image" /></td>
</tr>
<tr>
<td>4.3</td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>4.9</td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>Turbidity</td>
<td>Normalized Simulation Result</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td>5.5</td>
<td></td>
</tr>
<tr>
<td>6.1</td>
<td></td>
</tr>
<tr>
<td>6.7</td>
<td></td>
</tr>
<tr>
<td>Turbidity</td>
<td>Normalized Simulation Result</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------------</td>
</tr>
<tr>
<td>7.3</td>
<td><img src="image1.png" alt="Image" /></td>
</tr>
<tr>
<td>7.9</td>
<td><img src="image2.png" alt="Image" /></td>
</tr>
<tr>
<td>8.5</td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
</tbody>
</table>
External Simulations

Running External Simulations on the Same Computer

With external simulations, simulations can run on the same computer, keeping SPEOS CAA V5 Based available.

To do it, you can use the External Update function or the SPEOS Core application.

Running External Simulations Using External Update

1. From the specification tree, select a simulation.

2. Click External Update.

   The simulation is automatically isolated in the specification tree and exported.
   A .sv5 file is created in the SPEOS isolated files folder, without the simulation's results.
   The SPEOS Core application appears with the exported project opened.
   The new simulation appears in the specification tree.

3. From the SPEOS Core specification tree, select the simulation to update.
4. Click Local Update. Error XX% corresponds to the evolution of the total error number during the simulation.

- Or -

1. From the specification tree, select a simulation.

2. Click Isolate and Export (Simulations) or Export (Simulations). A .sv5 file is created in the SPEOS isolated files folder, with the simulation's results.

3. Click SPEOS Core (Tools).

4. Click File, Open..., and then select your .sv5 file.

5. Click Local Update. Error XX% corresponds to the evolution of the total error number during the simulation.

Parameterizing the SPEOS Core Application

The configuration file is located in folders.

- If using Windows Vista operation system, the configuration file is located in C:\Users\username\AppData\Roaming\OPTIS\SPEOS_CAA_V5_Based_VX\SPEOS_CAA_V5_Based.cfg.

- If using Windows XP operation system, the configuration file is located in C:\Documents and Settings\username\Application Data\OPTIS\SPEOS_CAA_V5_Based_VX\SPEOS_CAA_V5_Based.cfg.

1. Close SPEOS Core.exe.

2. Replace the following line:

   `<Thread-Mode>Thread-Mode-Automatic-Forced</Thread-Mode>`

   by the both following line:

   `<Thread-Mode>Thread-Mode-Value-Forced</Thread-Mode>`

3. Replace the 3 value by the wanted thread number.

   `<Thread-Number>3</Thread-Number>`

   Also note that if instead of:

   `<Thread-Mode>Thread-Mode-Value-Forced</Thread-Mode>`

   you must write down:

   `<Thread-Mode>Thread-Mode-Value-As-Default</Thread-Mode>`

When exporting, SPEOS Core.exe run on the thread number specified in the options, and saved in the .sv5 file.

Running a Simulation when Using OPTIS Distributed Computing

With OPTIS Distributed Computing, you can update a direct simulation or an inverse simulation over the network. To do it, you can use the Network Update function and manage the simulation running with the Simulation Spoolers Status application.

Running a Simulation Using Network Update

1. From the specification tree, select a simulation.

2. Click Network Update (Update). The simulation is automatically isolated in the specification tree and exported. A .sv5 file is created in the SPEOS isolated files folder, without the simulation's results.

3. Open the Simulation Spooler Status application (Tools) to check and manage your simulation progress.

   Note that you can also open the simulation spooler status application from the start menu, by clicking All Programs, OPTIS, Distributed Computing, Simulation Spoolers Status.

   - Or -

1. From the specification tree, select a simulation.
2. Click Isolate and Export (Simulations) or Export (Simulations).  
   A .sv5 file is created in the SPEOS isolated files folder, with the simulation's results.

3. Click SPEOS Core (Tools).
4. Click File, Open..., and then select your .sv5 file.
5. Click Network Update (Update).
6. Open the Simulation Spooler Status application (Tools) to check and manage your simulation progress.  
   Note that you can also open the simulation spooler status application from the start menu, by clicking All Programs, OPTIS, Distributed Computing, Simulation Spoolers Status.

- Or -
1. From the specification tree, select a simulation.
2. Click Isolate and Export (Simulations) or Export (Simulations).  
   A .sv5 file is created in the SPEOS isolated files folder, with the simulation's results.
3. In the SPEOS isolated files folder, right-click the .sv5 file, and then select Spool.
4. Open the Simulation Spooler Status application (Tools) to check and manage your simulation progress.  
   Note that you can also open the simulation spooler status application from the start menu, by clicking All Programs, OPTIS, Distributed Computing, Simulation Spoolers Status.

**Checking the Simulation Progress**

The Simulation Spooler Status is running within the user session and checks information every 5 seconds. However, to avoid network overbooking, spooler and servers are communicating every 30 seconds.

You must keep the application opened during the simulation.

- In the Simulation spoolers group box, you can find the following information about the Simulation Spoolers defined in the Configuration menu.
  - The Number of servers line displays the number of running simulation servers connected to the simulation spooler.  
    Note that for more information a tooltip displays the Simulation Servers related to the Simulation Spooler selected.
    The paused server are taking account into the Number of server(s).
  - The Simulation line displays information about the simulation status.
    Spooled means that the simulation is on hold until its turn.
    Simulating means that the simulation is running on each simulation server.
    Finished means that the simulation is completed. The results are available on the client computer.
    Aborting means that simulation is aborted.
    Stopping means that the simulation is stopping on each simulation server.
    The results are then uploaded and merged on the simulation spooler.
  - The Computer, User and Date lines display more information about the simulation parameters.
  - In the Simulation progress group box, you can find information about the Distributed Computing simulation running on a selected simulation spooler.
    - The Progress line displays the total achievement of the running simulation.
    - The Waiting the end of each simulation server line displays the distributed computing simulation progress since the launch date.
    - The Servers section displays the list of servers involved in the simulation.
      The servers names and simulation status are displayed.
   Simulation status displays the achievement percentage of the simulation on a server.
Note that sometimes it takes some time for the display's upgrade.

Troubleshooting: It can happen an error value appears. For more details, you can view system error codes (http://msdn.microsoft.com/en-us/library/windows/desktop/ms681382(v=vs.85).aspx).

At the end of the simulation, when the Simulation Spoolers Status is running, the Recently finished simulation dialog box appears to indicate which simulation is finished. The results are available on the client computer.

Managing the Simulation within the OPTIS Distributed Computing

From the Simulations menu, you can manage the simulations.

- You can click Stop the selected simulation to stop the selected running simulation on each simulation server.
  
  For direct and inverse simulation using the Monte Carlo algorithm, the obtained results are uploaded and merged.

- You can click Stop selected and merge the existing results to stop the selected running simulation on each simulation server.
  
  For direct and inverse simulations using the Monte Carlo algorithm, results of finished simulations are uploaded and merged.

- You can click Remove the selected simulation to remove the selected spooled simulation.

- You can click Abort the selected simulation so that the selected simulation server stops simulating and is restarted.
  
  No intermediate results are computed.

- You can click Clean the historic to clean the history.

- You can click Move up/Move down the selected simulation to reorder the simulation list, and then giving it a higher or lower priority.

  From the Configuration menu, you can add or remove simulation spooler to simulation spoolers list.

  In the Preferences group box, you can select the Display Recently finished simulation window to show a windows displaying the recently completed simulations.

Note that the simulation client always launches distributed computing simulation on the first simulation spooler of the list.

Managing the Servers within the OPTIS Distributed Computing

You can right-click a server in the servers' list in the Simulation spooler status to manage this server individually.

You can use the Stop Simulation on this Server, Abort Simulation on this Server and Kill Simulation on this Server options only when a simulation is running.

- You can click Stop Simulation on this Server to stop the simulation running on the server.
  
  For direct and inverse simulation using the Monte Carlo algorithm, the obtained results are uploaded and merged.

- You can click Abort Simulation on this Server to stop the simulation running on the server.

  If there are results they are not uploaded.

- You can click Kill Simulation on this Server to abort the simulation process.

  You can use this option when you notice that a server has not enough memory. You can see this by checking the CPU usage percentage, which starts to decrease, or by checking the remaining available RAM displayed.

  You can click Deactivate Server or Activate Server to activate or deactivate the server.

  The command is taken into account at the next simulation only.

Results

The simulations' results are automatically saved in a new folder generated after the first simulation run. This folder is called SPEOS output files and is created in the working directory of the project.
**Reading the HTML Report**

With HTML Report, you can read the analysis report. HTML Reports appear in the specification tree.

- In the Time analysis section, you can find information about the simulation time.
- In the Simulation parameters section, you can find information about the parameters set for the simulation.
- In the Error Report section, you can find the total number of errors reported in the simulation.
- In the Results section, you can find information about the simulation result.
- In the Geometry Report section, you can find information about geometries.
- In the Body List section, you can find information about faces with their applied face optical properties.

When a Luminaire Source is used, information is added in the simulation report. In the Luminaire Wattage section, you can find the number of luminares.

**XMP Result**

**Visualizing a XMP Result**

XMP results appear in the specification tree.

- You can click Customize View Parameters and Material to visualize the XMP result in the 3D view.

**Direct simulation result**

**Inverse simulation result**

- From the specification tree, double click the .xmp file to open it using the Virtual Photometric Lab.

Note that the visualization within SPEOS CAA V5 Based software is managed from the Virtual Photometric Lab or the Virtual Human Vision Lab.

**Meridian/parallel orientation**

**Conoscopic orientation**
Creating a Measure from a XMP Result

With Measure, you can create a measure from a XMP file.

1. Click Measure (Measure).
2. Set the parameters (see page 160).
   You can click Preview to preview the measure.
3. Click OK.

   The measure appears in the specification tree, in the Measures section.

Parameters of a Measure

- In the XMP box, you must select a .xmp file from the specification tree.
- From the list, you must select the measure type.
  - If you select Point, you must set the Inputs parameters.
    From the Color System list, you can select a colorimetric data.
  - For more details, you can view Colorimetric Data.
    You must set a value in the X and Y spinboxes.
  - If you select Line, you must set the Inputs parameters.
    You must set a value in the Sampling spinbox.
    You must set a value in the X1, X2, Y1, and Y2 spinboxes.
  - If you select Surface, you must select a type from the list and set the Inputs parameters.
    If you select the Rectangle type, set the value of the Center X, Center Y Height and Width spinboxes.
    If you select the Ellipse type, set the value of the Center X, Center Y, Radius X and Radius Y spinboxes.
    If you select the Polygon type, in the Point number box, you must type the number of points needed to define
    the polygonal area. In the Pn X and Pn Y boxes, you must type values or change them by using the arrows.

   Note that if an intensity sensor is involved in the simulation result, you must set the angle parameters.
   For Surface, Rectangle, you must set the Inputs parameters, in the Phi Max, Phi Min, Theta Max and Theta Min
   boxes.

   With the Rectangle type, you can click Local Update, open the .xmp file, click Surface/Section, click
   Colorimetric data. You can view that Surface/Section values are related to local measure belong to the color and view the correlation
   regarding values in the specification tree.

LPF Result

Visualizing a LPF Result for Interactive Simulation

LPF results appear only when using the light expert in the simulation.

1. Expand the Interactive Simulation node.
2. Double-click the .lpf file.
3. In the Required faces or bodies group box, you must select faces or bodies by clicking them in the specification
   tree or in the graphics area.
   The rays impacting the required faces or bodies are displayed in the 3D view.
4. In the Rejected faces or bodies group box, you must select faces or bodies by clicking them in the specification
   tree or in the graphics area.
   The rays, excepted those impacting the rejected faces or bodies are displayed in the 3D view.
Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection.... Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

No rays may be traced when selecting a body and keeping the AND mode. Indeed, the rays need to hit all faces of the body to be represented in the ray tracing.

5. In the Mode box, you must select a mode from the list.
   - If you select AND, rays with one intersection with the each face of the selected faces list are displayed.
   - If you select OR, rays with almost one intersection with one of the selected faces list are displayed.

6. In the Number of rays box, you must type or edit a value.
   - This parameter sets the number of displayed rays during the interactive ray tracing.

7. By clicking or Export Simulation's Rays as Geometry (Simulations), you can export LXP rays as geometry.
   - For more details, you can view Export Simulation's Rays as Geometry (see page 106).

**Visualizing a LPF Result for Direct or Inverse Simulation**

LPF results appear only when using the light expert in the simulation.

1. From the simulation node, expand the .xmp node.
2. Double-click the .lpf file.
3. In the Required faces or bodies group box, you must select elements by clicking them in the specification tree or in the graphics area, or using the Other Selection option (see page 11):

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

The rays impacting the required faces and reaching the sensor are displayed in the 3D view.

4. In the Rejected faces or bodies, you must select elements by clicking them in the specification tree or in the graphics area, or using the Other Selection option (see page 11):

<table>
<thead>
<tr>
<th>Faces</th>
<th>Surfaces</th>
<th>Solids</th>
</tr>
</thead>
</table>

The rays reaching the sensor, excepted those impacting the rejected faces or bodies, are displayed in the 3D view.

- Note that you can also select an entire body. To do so, you must highlight a face of the geometry, right-click on it and select Other Selection....
  - Then, in the specification tree, you must select the body corresponding to the selected face. This body is then added to the selection list.

- Note that no rays may be traced when selecting a body and keeping the AND mode. Indeed, the rays need to hit all faces of the body to be represented in the ray tracing.

5. In the Mode box, you must select a mode from the list.

- If you select AND, rays with one intersection with the each face of the selected faces list are displayed.
- If you select OR, rays with almost one intersection with one of the selected faces list are displayed.

6. You must set a value in the Number of rays spinbox.
This parameter sets the number of displayed rays during the interactive ray tracing.

7. By clicking \(\text{ Export Simulation's Rays as Geometry }\) (Simulations), you can export LXP rays as geometry.

Visualizing a Ray Result

Ray file results appear in the specification tree when using the ray file in a direct or inverse simulation.

• From the specification tree, you can double click the .ray file to open it using the Ray File Editor.

Visualizing an Eulumdat, IESNA LM-63, OPTIS Intensity Distribution Result

Using an interactive or direct simulation including an intensity sensor creates a .ldt, .ies, .intensity or .int file result.


• From the specification tree, you can double click the .ldt file to open it using the Eulumdat Viewer.

• From the specification tree, you can double click the .ies file to open it using the IESNA LM-63 Viewer.

• From the specification tree, you can double click the .intensity or .int file to open it using the OPTIS Intensity Viewer.

Visualizing an Unpolished, Anisotropic BSDF or Complete Scattering Result

Using a virtual BSDF bench simulation creates an .anisotropicbsdf, .brdf or .unpolished file result.

Anisotropic BSDF, complete scattering or unpolished result appear in the specification tree.

• From the specification tree, you can double click the .anisotropicbsdf, .brdf or .unpolished file to open it using the BSDF - BRDF - Anisotropic Surface Viewer.

Visualizing a XM3 Result

XM3 results appear in the specification tree.

1. From the specification tree, double-click the .xm3 file to open it with the Virtual 3D Photometric Lab.
2. From the Display box, select Mapping from the list to display the XM3 Result.

XM3 result

Results Management

Managing the Results within the Specification Tree
All the results appear in the specification tree.
You can right-click on the result to manage it.
- If you want to focus on a result in the specification tree, click Center graph.
- If you want to focus on a result in the graphic area, click Reframe On.
- If you want to hide or display the result, click Hide/Show.
- If you want to edit the result properties, click Properties.

For more details, you can view Results Properties (see page 165).

Managing the Results Sub-Tree
To open a result sub-tree from the specification tree, you can right-click on a result and click Open Sub-Tree.
- From LXP, you can select true or false.
- From Number of rays, you can type a value or change it by using the arrows.

Managing the Results Objects
By right-clicking on a result from the specification tree, you can click Result Name object.
- If you want to edit the result definition, click Definition....
- If you want to update a result, click Local Update.
- If you want to update a series of results without blocking the process if one of them leads to an error, click Local Update Without Notification.
- If you want to activate or deactivate a component, click Activate / Deactivate Component.
- If you want to move up and down the result in the Simulations section of the specification tree, click Move up or Move down.
- If you want to open a .xmp file with a viewer, click Virtual Human Vision Lab.
- If you want to directly open the folder where an isolated simulation is located, click Open folder.

The Open folder option is available for isolated simulations only.

Isolating a Simulation Result
You can isolate a simulation result in the specification tree, independently of the historic update. This type of simulation gives an easy access to results files. You can use it when a result is good and you are not sure if you can optimize it.

Isolated Simulations have no link with sources, geometries and sensors.
Isolated simulations appear in the same node than the simulation node.
1. In the specification tree, select the simulation you want to isolate.

2. Click Isolate (Simulations).
   
   A new simulation is added in the specification tree.

   The Isolated simulation inherits its name from the original simulation followed by a dot and the index of the isolation.

   This new simulation gives an easy access to results files. You can right-click on the isolated simulation to open them.

   The isolated simulation cannot be updated or run anymore.

**Results Properties**

By right-clicking on a result from the specification tree, you can click Properties to edit results properties.

**Setting the Parameters**

You must click More… to edit the Parameters tab.

- You can select the Show results in 3D check box.
  
  With this parameter, you can display the results in the 3D view.

- You can select the Show null values of results as transparent in 3D check box.
  
  With this parameter, you can display the results with an advantage of image transparency for null values.

Note that if you click Tools, Options…, and edit the General tab, you can select these two parameters. If you do so, you set the default value of these parameters, which is used during the first update of the simulation, and create the results features.

- If an intensity sensor has been included in the simulation, the Visualization distance box appears.
  
  You can set a value in the spinbox.

  In case of a near field intensity sensor, the cell distance value is given to the visualization distance value. For more details, you can view Parameters of an Intensity Sensor (see page 73).
Creating a Light Modeling System

You must have the S_SV5_LM2, S_SV5_LM3 or S_SV5_LM4 solution.

With this tutorial, you are about to learn the basics Light Modeling features.

Lesson 1: Opening Project
2. Open SPEOS CAA V5 Based VXX.
3. Click Start, Analysis & Simulation, Light Modeling.
4. From CATIA’s tree, right-click Product1, and then select Components, Existing Component....
5. Open the Guide.CATPart file.
6. From CATIA’s tree, right-click Product1, and then select Properties.
7. In the Product group box, in the Part Number, type GS_LM. The product name is modified in CATIA’s tree.
8. Click File, Save As.

Lesson 2: Applying Materials to a Product
1. From CATIA’s tree, expand the Guide (Guide.1) node, and then expand the Guide node.
2. Select the Guide body.
3. Click Apply Material (Optical Properties).
4. From the Library dialog box, in the Other tab, select the Plexiglass material.
5. Click OK. The material appears in CATIA’s tree, in the Guide body section.
6. From CATIA’s tree, select the LED body.
7. Click Apply Material (Optical Properties).
8. From the Library dialog box, in the Other tab, select the Plastic material.
9. Click OK. The material appears in CATIA’s tree, in the LED body section.

Lesson 3: Adding Optical Properties to Materials
1. From CATIA’s tree, select Plexiglass, and click Edit SPEOS properties.
2. In the Volume optical properties (VOP) group box, select Optic from the Type list. You must not change the Index, Absorption, and Contringence values.
3. In the Surface optical properties (SOP) group box, select Optical polished from the Type list.
4. Click Close.
5. From CATIA’s tree, select Plastic, and click Edit SPEOS properties.
6. In the Volume optical properties (VOP) group box, select Opaque from the Type list.
7. In the Surface optical properties (SOP) group box, select Mirror from the Type list.
8. In the Reflectance box, type 0.
9. Click Close.

**Lesson 4: Creating an Interactive Source**

1. Click Interactive Source (Sources).
2. In the Start group box, in the Point box, click the Vertex/Sketch.1 point in the graphics area.
3. In the End group box, in the Curve box, click the Edge/Sketch.1 line in the graphics area.
4. In the Sampling box, type 30.
5. Click OK.

   The interactive source appears in the specification tree.
Lesson 5: Creating an Interactive Simulation

1. Click Interactive Simulation (Simulations).
2. In the Sources box, select Interactive source.
3. In the Geometries box, select the Guide body.
4. Click OK.

The interactive simulation appears in the specification tree.

The interactive simulation appears in the 3D view.

Lesson 6: Creating a Surface Source

1. Click Surface Source (Sources).
2. In the Flux group box, select Radiant, then in the Flux box, type 1.
3. In the Emissive faces box, click the Face/Pad.4 face in the graphics area.

![Image of Emissive faces with Face/Pad.4 highlighted]

4. In the Intensity group box, in the Total angle box, type 15.
5. Click OK.
6. The surface source appears in the specification tree.
   The surface source appears in the 3D view.

![Image of surface source in 3D view]

**Lesson 7: Creating an Irradiance Sensor**

1. Click Irradiance Sensor (Sensor).
2. Set the Type to Photometric.
3. In the Axis system group box, in the Origin box, click the Vertex/Sketch.1 point in the graphics area.

4. In the X direction box, click the Edge/Pad.1 line in the graphics area.

5. In the Y direction box, click the Edge/Pad.1 line in the graphics area.

6. Select Planar from the Integration type list.

7. In the X group box, type -5 in the Start box, and type 5 in the End box.
8. In the Y group box, type -5 in the Start box, and type 5 in the End box.
9. Click Reverse direction in the Integration direction line.
10. Click OK.
   The irradiance sensor appears in the specification tree.
   The irradiance sensor appears in the 3D view.

Lesson 8: Creating a Direct Simulation

1. Click Direct Simulation (Simulations).
2. In the Sources box, select Surface source.1.
3. In the Geometries box, select the Guide body.
4. In the Sensors box, select Irradiance sensor.1.
5. Click OK.
   The direct simulation appears in the specification tree.
6. From the specification tree, select Direct simulation.1, and then click Local Update (Update).
The direct simulation appears in the 3D view.

7. Expand the Direct simulation.1 node to access to the results.

**Lesson 9: Analyzing Results**

1. From the specification tree, expand the Direct simulation.1 node.
2. Double click the .xmp file to open the result.
3. Click Surface / Section.
4. From the Shape list, select Horizontal to access the uniformity analysis.

**Applying VOP on Surface**

You must have the S_SV5_LM1, S_SV5_LM2, S_SV5_M3 or S_SV5_LM4 solution.

10 minutes

2. From your recent created folder, open the VOPOnSurface.CATProduct file.

3. Click Interactive Simulation (Simulations).

4. In the Sources box, select Beam.

5. In the Geometries box, select Join.Prism and Screen.

6. Click Preview.
   The following warning appears:
   "Invalid input parameters
   VOP on Surface parameter should be set to true to apply Volume Optical Properties (VOP) on a surface
   (VOPOnSurface/Prism.1/Prism/Open.Prism/Join.Prism)
   Set VOP on Surface parameter to true"

7. Click OK.
   Note that Volume Optical Properties were applied to a surface body.
   SPEOS CAA V5 Based displays a warning because this situation lead to propagation error when the surface body
   is not closed.

8. In the Geometries box, right-click Join.Prism, and then select Clear selection.

9. Click OK.

10. From the specification tree, expand the Simulations node.

11. Right-click Interactive simulation.1, and then select Properties.

12. Click More... to edit the Simulation tab.

13. From the VOP on surface list, select true.

14. Click OK.

15. From the specification tree, double-click Interactive simulation.1.


17. Click OK.
   The volume optical properties appear in the 3D view.

Note that the propagation of rays is done in the prism, even if it is not a closed geometry.

Creating a Ray File Source
You must have the S_SV5_LM1, S_SV5_LM2, S_SV5_LM3 or S_SV5_LM4 solution.

10 minutes

1. Copy SV5_Tutorials_RayFileSource_R17V11.zip

2. From your recent created folder, open the RayFileSource.CATProduct file.

3. Click Ray file Source (Sources).

4. In the Axis system group box, in the Origin box, click the Point.Origin point in the graphics area.

5. In the X Direction box, click the Line.X line in the graphic area.
6. In the Y Direction box, click the Line.Y line in the graphics area.

7. In the Photometry group box, in the Ray file name, browse the Rays.ray file.
8. In the Exit Geometries group box, select Bulb.
9. Click OK.

   The bulb selection modifies the start position of the rays but not the direction.
   The ray file source appears in the specification tree.
   The ray file source appears in the 3D view.

10. Click Interactive Simulation (Simulations).
11. In the Sources box, select Ray file source.1.
12. In the Geometries box, select the Filament and Reflector bodies.
13. Click OK.

   The interactive simulation appears in the specification tree.
The interactive simulation appears in the 3D view.

The bulb is taken into account during the propagation, excepted at the emission step.

Creating an Integration Direction

You must have the S_SV5_LM2, S_SV5_LM3 or S_SV5_LM4 solution.

15 minutes


2. From your recent created folder, open the IntegrationDirection.CATProduct file.

3. Click Irradiance Sensor (Sensors).

4. Set the Type to Photometric.

5. In the Axis system group box, in the Origin box, click the Vertex/Point.Origin point in the graphics area.
6. In the X direction box, click the Edge/Line.X line in the graphics area.

7. In the Y direction box, click the Edge/Line.Y line in the graphics area.

8. For both X and Y group boxes, type 50 in the Sampling boxes.

9. Click OK.

The irradiance sensor appears in the specification tree.
The irradiance sensor appears in the 3D view.
10. From the specification tree, right click Irradiance sensor.1 and then select Properties.
11. In the Feature Name box, type Orthogonal.
12. Click OK.
   The irradiance sensor name is modified in the specification tree.
13. Click Irradiance Sensor (Sensors).
14. Repeat steps 4 to 8.
15. In the Integration direction box, click the Edge/Line.Integration line in the graphics area.
16. Click OK.
   The irradiance sensor appears in the specification tree.
   The irradiance sensor appears in the 3D view.
17. From the specification tree, right-click Irradiance sensor.2 and then select Properties.
18. In the Feature Name box, type Oriented.
19. Click OK.
   The irradiance sensor name is modified in the specification tree.
20. Click Direct Simulation (Simulations).
21. In the Sources box, select Light.
22. In the Sensors box, select Orthogonal and Oriented.
23. In the Number of rays box, type 1000000.
24. Click OK.
25. From the specification tree, right click Direct simulation.1 and then select Properties.
26. In the Feature Name box, type LM.
27. Click OK.
   The direct simulation name is modified in the specification tree.
28. From the specification tree, select LM, and then click Local Update (Update).
29. Double-click the .xmp files to open the results.
30. For both results click Illuminance , and compare the value.

Note that the ratio between illuminance level is the cos of the incidence angle, \( \cos(60\text{deg}) = 0.5 \)

**Running a Light Modeling Optimization**

*You must have the S_SV5_LM2 solution with the O_SV5_OPTIM2 option.*

30 minutes

2. From your recent created folder, open the Optimization_LM.CATProduct file.

3. From the specification tree, expand the Simulations node.

4. Select Direct simulation.1 and then click Local Update (Update).

5. From the specification tree, double-click the .xmp file to open the result.

6. Click Surface / Section.

7. From the Section group box, select Horizontal, and then check the spot.

8. Close the result.

9. Click Measure (Measure).

10. In the XMP box, select the .xmp file.

11. From the list, select Line.

12. In the Sampling box type 1000.
13. In the P1 X box type 0, and then in the P2 X box type 50.
14. Click Preview.
15. From the Outputs list, select X Max.
16. Click OK.
17. Click Start, Knowledgeware, Product Engineer Optimizer.
18. Click Optimization (Product Engineering Optimizer).
19. In the Optimized parameter group box, click Select....
20. From the list select `SPEOS CAA V5 Based\Measures\Measure.1\X Max`.
21. Click OK.
22. In the Free Parameters group box, click Edit list.
23. From the Parameters list, double-click `Prism\Input.Angle`.
24. Click OK.
25. In the Free Parameters group box, click `Prism\Input.Angle`, and then click Edit ranges and step.
26. In the Inf. Range box type 35.
27. In the Sup. Range box type 65.
28. In the Step box type 1.
29. Click OK.
30. In the Termination criteria group box, in the Maximum number of updates box, type 20.
31. Click to clear the Consecutive updates without improvements and the Maximum time (minutes) check boxes.
32. Click Run optimization.
33. Save the results in a .txt format.
   The optimization process starts.
34. From the Computations results tab, click Show curves....

35. In the Sorted results group box, select the best value, and then click Apply values to parameters.
36. Click OK.
The optimization appears in the 3D view.

37. From CATIA's tree, expand the Parameters node, and then check if the Measured.Index value matches the refractive index of the prism.

Including a 3D Texture in an Optical System

You must have the S_SV5_LM2 or S_SV5_VE2 solution with the O_SV5_3DTEXT option.

30 minutes

Lesson 1: Creating a Map

1. Open SPEOS CAA V5 Based.
2. Click Tools, Macro, Macros....
3. Click Macro libraries....
4. From the Library type list, select VBA projects.
5. Click Create new library....
6. Type the path and the name of the VBA project, and then click OK.
   The new library is added to the list.
7. Click Close.
8. Click Create.
9. Click OK to validate the macro name.
10. Click Edit....
   The Visual Basic Editor dialog box appears.
11. Start to write a Visual Basic for Application code as follow:

```vba
Sub CATMain()
    Dim fso As Object
    Set fso = CreateObject("Scripting.FileSystemObject")
    Dim ftextfile As Object
    Set ftextfile = fso.CreateTextFile("C:\MyDistribution.OPT3DMapping", True)
End Sub
```
This code represents the sub routine CATMain that creates a text file in the C directory, named MyDistribution.OPT3DMapping.

An example of this file can be found in MyDistribution.zip (http://portal.optis-world.com/documentation/UG/SV5/ZIP/MyDistribution.zip).

12. Continue the code by writing a double loop that increments the X and Y positions of the 3D texture's patterns. This loop starts from the (0, 0) position and uses a spacing of 0.2 mm between the patterns.

```vba
Sub CATMain()
    Dim fsao As Object
    Set fsao = CreateObject("scripting.filesystemobject")
    Dim ftextfile As Object
    Set ftextfile = fsao.createtextfile("C:\MyDistribution.OPT3DMapping", True)
    ftextfile.writeline 50
    For i = 0 To 0.9 Step 0.2
        For j = 0 To 1.9 Step 0.2
            ftextfile.writeline i & " " & j & " 0 1 0 0 1 0 1 1 1"
        Next j
    Next i
    ftextfile.Close
End Sub
```

13. Press F5 to run the macro.

   Note that you can open the pattern distribution map by using Notepad.

**Lesson 2: Creating a 3D Texture**


2. From your recent created folder, open the 3DTexture.CATProduct file.

3. **Click 3D Texture** (Optical Properties).

4. In the Axis System group box, in the Origin box, click the Vertex/3Dtexture.origin point in the graphics area.
5. In the X Direction box, click the Edge/X.axis line in the graphics area.

![Image of X Direction]

6. In the Y Direction box, click the Edge/Y.axis line in the graphics area.

![Image of Y Direction]


8. In the Mapping definition box, select From file and browse the MyDistribution.OPT3DMapping file, located in the C: drive.

9. In the Pattern group box, in the Part file box, browse the Pattern.CATPart file.

10. Select the Preview meshing check box.
    
    Note that only the mapping axes are displayed if you do not select the Preview meshing check box.
The patterns appear in the 3D view.

11. In the Scale box, type 0.1 to decrease the patterns size.

Note that you can tune the preview by clicking the arrows and moving the mouse along its direction.
12. From the Zone selection section, select the Sphere check box.

13. Click on OK.
   The 3D texture appears in the 3D view.
   The 3D texture appears in the specification tree.

**Lesson 3: Running an Interactive Simulation**

1. From the specification tree, double-click Interactive simulation.
2. In the Geometries box, add 3D Texture.
3. Click OK.
The interactive simulation appears in the 3D view.

**Lesson 4: Running a Direct Simulation**

1. From the specification tree, double-click Direct simulation.1.
2. In the Geometries box, add 3D Texture.1.
3. Click OK.
4. From the specification tree, select Direct simulation.1 and then click Local Update (Update).
5. Click Level and set the optimal maximum value in the Value box.
6. Double-click the .xmp files to open the results.
Creating a Backlight Unit Optimization

You must have the S_SV5_LM2 solution with the O_SV5_3DTEXT and O_SV5_OPTIM1 options.
You must have CATIA R19 version or higher.

Lesson 1: Opening the Project
1. Open SPEOS CAA V5 Based VXX.
2. Click Tools, Options..., General, Parameters and Measure to edit the Knowledge tab.
3. Select the With value and With formula check boxes.
4. From the Knowledge Environment tab, select the Load extended language libraries and All packages check boxes.
5. From the tree, expand the Infrastructure node, and then select Part Infrastructure.
6. From the Display tab, select the Constraints, Parameters and Relations check boxes.
7. Click OK.
9. From your recent created folder, open the Backlight_unit.CATProduct file.

Lesson 2: Creating Parameters
1. Click Tools, Formula....
2. Click New parameter of type.
3. Select Real type with Single Value.
4. Rename it param_x_min and set its value to -24.
5. Repeat these steps to create the following parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>param_x_max</td>
<td>Real</td>
<td>24</td>
</tr>
<tr>
<td>param_y_min</td>
<td>Real</td>
<td>0</td>
</tr>
<tr>
<td>param_y_max</td>
<td>Real</td>
<td>71</td>
</tr>
<tr>
<td>param_pitch_min</td>
<td>Real</td>
<td>0.033</td>
</tr>
<tr>
<td>param_pitch_max</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch1</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch2</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch3</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch4</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch5</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch6</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch7</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch8</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_raynb</td>
<td>Integer</td>
<td>10,000,000</td>
</tr>
<tr>
<td>targetflux</td>
<td>Luminous flux</td>
<td>0</td>
</tr>
</tbody>
</table>

6. Click OK.
   The parameters appear in the CATIA's tree, in the Parameters section.
Lesson 3: Creating a VB Script

1. Click Start, Knowledgeware, Knowledge Advisor, and then click Macros with arguments (Actions).
2. In the Definition group box, in the argument(s) box, type the name of all the created parameters, separating each other by a comma.
3. Type the VB code by following the steps.
   - Start the VB script by typing the following line to set the arguments.
     ```vb
     Sub main(param_x_min, param_x_max, param_y_min, param_y_max, param_pitch_min,
               param_pitch_max, param_pitch1, param_pitch2, param_pitch3, param_pitch4,
               param_pitch5, param_pitch6, param_pitch7, param_pitch8, param_raynb, targetflux)
     ```
   - Continue with the following lines to make a reference to the active document.
     ```vb
     Dim productDocument1 As Document
     Set productDocument1 = CATIA.ActiveDocument
     Dim product1 As Product
     Set product1 = productDocument1.Product
     ```
   - The following code changes the number of rays of the direct simulation in order to indicate its status is not up to date and therefore the simulation needs to be run.
     ```vb
     param_raynb.Value = 1000
     product1.Update
     param_raynb.Value = 1000000
     product1.Update
     ```
   - Then, type a File System Object in order to create a temporary text file that includes the 3D texture distribution.
     ```vb
     Dim fso As Object
     Set fso = CreateObject("Scripting.FileSystemObject")
     Dim f As Object
     Set f = fso.CreateTextFile(CATIA.ActiveDocument.Path & "\HalfSphere_dist_temp.txt", True)
     ```
   - Some variables have to be defined and initialized.
     - IsOdd represents the flag indicating whether a vertical line of the 3D texture distributed is odd or pair.
       ```vb
       Dim IsOdd As Boolean
       IsOdd = True
       ```
     - dot_spacingX and dot_spacingY indicate respectively the spacing along the X and Y directions.
     - dot_posX and dot_posY indicate respectively the dot position along the X and Y directions.
     - dot_num indicates the number of dots.
     - NumbAreas represents the number of areas.
     - areaNumber represents the actual area number.
     ```vb
     Dim dot_spacingX, dot_spacingY, dot_posX, dot_posY, dot_num, NumbAreas, areaNumber
     dot_spacingY = 0
     dot_posY = 0
     dot_num = 0
     NumbAreas = 8
     areaNumber = 1
     dot_spacingX = param_pitch1.Value
     ```
   - Do While loop increments the value of the dot_posY parameter until it reaches the maximum length defined by param_y_max.Value - param_pitch_min.Value.
     ```vb
     Do While dot_posY < (param_y_max.Value - param_pitch_min.Value)
     ```
- Type the code of the area number.
  
  3 different cases are distinguished between the first area, the final one and the others in order to initialize the value of the \textit{areastart} and \textit{areaend} parameters corresponding to the starting and ending length of the area.

  \begin{verbatim}
  If \textit{areaNumber} = 1 Then
    \textit{areastart} = \textit{param\_pitch\_min.Value}
    \textit{areaend} = \textit{areaNumber} * \textit{param\_y\_max.Value} / \textit{NumbAreas}
  ElseIf \textit{areaNumber} = 8 Then
    \textit{areastart} = (\textit{areaNumber} - 1) * \textit{param\_y\_max.Value} / \textit{NumbAreas}
    \textit{areaend} = \textit{param\_y\_max.Value} - \textit{param\_pitch\_min.Value}
  Else
    \textit{areastart} = (\textit{areaNumber} - 1) * \textit{param\_y\_max.Value} / \textit{NumbAreas}
    \textit{areaend} = \textit{areaNumber} * \textit{param\_y\_max.Value} / \textit{NumbAreas}
  End If
  \end{verbatim}

- Then, a test is made on the \textit{dot\_spacing\_Y} parameter as it should not be lower than the minimum pitch value.

  \begin{verbatim}
  If \textit{dot\_spacing\_Y} < \textit{param\_pitch\_min.Value} Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch1.Value}
  End If
  \end{verbatim}

- The \textit{IsOdd} parameter is checked in order to initialize the dot position along the X direction. If the line is odd, it starts with the centred dot, with two symmetrical dots otherwise. The number of dots is incremented accordingly.

  \begin{verbatim}
  If \textit{IsOdd} = True Then
    \textit{dot\_pos\_X} = 0
    \text{f.writeline dot\_pos\_X & " " & \textit{dot\_pos\_Y} & " 0 1 0 0 0 1 0 1 1 1"
    \textit{dot\_numb} = \textit{dot\_numb} + 1
  ElseIf \textit{IsOdd} = False Then
    \textit{dot\_pos\_X} = \textit{dot\_spacing\_X} / 2
    \text{f.writeline dot\_pos\_X & " " & \textit{dot\_pos\_Y} & " 0 1 0 0 0 1 0 1 1 1"
    \text{f.writeline -dot\_pos\_X & " " & \textit{dot\_pos\_Y} & " 0 1 0 0 0 1 0 1 1 1"
    \textit{dot\_numb} = \textit{dot\_numb} + 2
  End If
  \end{verbatim}

- The line is continued until the dot position along the X direction reaches a maximum length value.

  \begin{verbatim}
  \text{Do While Abs(\textit{dot\_pos\_X}) <= (\textit{param\_x\_max.Value} - \textit{param\_pitch\_min.Value})
  \text{    \textit{dot\_pos\_X} = \textit{dot\_pos\_X} + \textit{dot\_spacing\_X}
    \text{f.writeline dot\_pos\_X & " " & \textit{dot\_pos\_Y} & " 0 1 0 0 0 1 0 1 1 1"
    \text{f.writeline -dot\_pos\_X & " " & \textit{dot\_pos\_Y} & " 0 1 0 0 0 1 0 1 1 1"
    \textit{dot\_numb} = \textit{dot\_numb} + 2
  \text{Loop}
  \end{verbatim}

- The \textit{dot\_pos\_Y} parameter is checked to see if the dot is still located in the corresponding area. If it is not the case, the area number is incremented.

  \begin{verbatim}
  \text{If dot\_pos\_Y} >= \textit{areastart And dot\_pos\_Y} < \textit{areaend Then
  \text{'nothing}
  \text{Else
    \textit{areaNumber} = \textit{areaNumber} + 1
  \text{End If}
  \end{verbatim}

- Then, the \textit{dot\_pos\_Y} parameter takes the value of the corresponding pitch according to the area number.

  \begin{verbatim}
  \text{If \textit{areaNumber} = 1 Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch1.Value}
  \text{ElseIf \textit{areaNumber} = 2 Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch2.Value}
  \text{ElseIf \textit{areaNumber} = 3 Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch3.Value}
  \text{ElseIf \textit{areaNumber} = 4 Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch4.Value}
  \text{ElseIf \textit{areaNumber} = 5 Then
    \textit{dot\_spacing\_Y} = \textit{param\_pitch5.Value}
  \end{verbatim}
ElseIf areaNumber = 6 Then
dot_spacingY = param_pitch6.Value
ElseIf areaNumber = 7 Then
dot_spacingY = param_pitch7.Value
ElseIf areaNumber = 8 Then
dot_spacingY = param_pitch8.Value
End If

- The dot position along the Y direction is then incremented. The dot_spacingX parameter is then modified according to the new value of the pitch along Y direction.
  
  dot_posY = dot_posY + dot_spacingY
  dot_spacingX = 2 * dot_spacingY / Sqr(3)

- Finally, the IsOdd flag is updated.
  
  If IsOdd = True Then
      IsOdd = False
  Else
      IsOdd = True
  End If

Loop

This concludes the “Do While” loop.

- The text file is closed. It is then open to be read.

  f.Close
  Set f = fso.OpenTextFile(CATIA.ActiveDocument.Path & "\HalfSphere_dist_temp.txt", 1)

- A new text file is created and corresponds this time to the actual 3D mapping file. Its first line corresponds to the number of dots.

  Dim ftemp As Object
  Set ftemp = fso.CreateTextFile(CATIA.ActiveDocument.Path & "\SPEOS input files\BLU_mapping.OPT3DMapping", True)
  ftemp.writeline dot_numb

- A loop then writes the content of the first text file in the second one. Both text files are then closed.

  Do While f.AtEndOfStream <> True
      ftemp.writeline f.ReadLine
  Loop
  ftemp.Close
  Set ftemp = Nothing
  f.Close
  Set f = Nothing
End Sub

1. Click OK to validate the script.
   The script appears in the CATIA’s tree, in the Relations section.
   The code of the macro is here.

Lesson 4: Creating a Reaction

1. Click Start, Knowledgeware, Knowledge Advisor, and then click Reactions (Reactive Features).
2. In the Sources box, select param_pitch1 to param_pitch8.
   param_pitch1 to param_pitch8 parameters have to be in the same order than in the VB script.
3. Click Edit action....
   The Action Editor: Reaction dialog box appears.
4. Double click VB Script1 in the tree.
5. From the Dictionary group box, select Messages and macros.
6. From the Members of Messages and macros group box, double-click VB Script -> Run.
7. In the Inputs box, select all the created parameters. Parameters have to be in the same order than in the VB script.

8. Click OK.

9. Click OK to validate the Reaction. The reaction appears in the CATIA's tree.

10. Test the reaction by changing the value of param_pitch1 for example. The result of the reaction is the .txt file creation.

Lesson 5: Writing the Optimization Script

You are about to generate from scratches a macro that runs an optimization loop following the 6 next steps.

You are about to learn how to:
- Access an existing SPEOS CAA V5 Based simulation feature and how to update it from a script.
- Control the OPTIS Virtual Photometric Lab application.
- Interact between CATIA V5, SPEOS CAA V5 Based and the OPTIS Virtual Photometric Lab.

Lesson 1: Creating the Macro

1. Click Tools, Macro, Macros....
2. Click Macro libraries....
3. From the Library type list, select VBA projects.
4. Click Create new library....
5. Enter your working directory BP_BLU_Optim1.catvba as name of the project.
6. Click OK.
7. Click Close.
8. Click Create....
9. Click OK.
10. Click Edit... to display the Visual Basic Editor.

Lesson 2: Declaring and Initializing Variables

1. Write the following variables before Sub CATMain() in order to define them as global variable and thus use them in procedure.

```vbscript
Dim fso As Object
Dim XMPViewer As Object

' CAA
Dim productDocument1 As ProductDocument
Dim product1 As Product

' Parameters
Dim param_pitch_min As RealParam
Dim param_pitch_max As RealParam
Dim param_pitch1 As RealParam
Dim param_pitch2 As RealParam
Dim param_pitch3 As RealParam
Dim param_pitch4 As RealParam
Dim param_pitch5 As RealParam
Dim param_pitch6 As RealParam
Dim param_pitch7 As RealParam
Dim param_pitch8 As RealParam
```
Dim targetflux As RealParam
' Simulation
Dim oPTSimuDirect1 As OPTSimuDirect
Dim oPTSpeosFeature1 As OPTSpeosFeature
Dim XMPfile As String
Dim AreaListCount
Dim simuNumber
Dim FolderPath As String

2. Declare the following variable that is useful for optimization algorithm:
   Global DotRep_Min As Double, DotRep_Max As Double, DotRep_Avg As Double
   Global DeltaEInf As Double, DeltaEMid As Double, DeltaESup As Double
   Global SimuInf As Boolean, SimuMid As Boolean, SimuSup As Boolean

3. Start the CATMain routine by writing the following lines to make a reference to the active document:
   Sub CATMain()
   Set productDocument1 = CATIA.ActiveDocument
   Set product1 = productDocument1.Product
   End Sub

4. A reference to each parameter is established using the following code:
   Dim parameters1 As Parameters
   Set parameters1 = product1.Parameters
   Set param_pitch_min = parameters1.Item("param_pitch_min")
   Set param_pitch_max = parameters1.Item("param_pitch_max")
   Set param_pitch1 = parameters1.Item("param_pitch1")
   Set param_pitch2 = parameters1.Item("param_pitch2")
   Set param_pitch3 = parameters1.Item("param_pitch3")
   Set param_pitch4 = parameters1.Item("param_pitch4")
   Set param_pitch5 = parameters1.Item("param_pitch5")
   Set param_pitch6 = parameters1.Item("param_pitch6")
   Set param_pitch7 = parameters1.Item("param_pitch7")
   Set param_pitch8 = parameters1.Item("param_pitch8")
   Set targetflux = parameters1.Item("targetflux")

5. A reference to the direct simulation needs to be created using references to the “SPEOS CAA V5 Based” node and the “Simulations” node:
   Dim oPTSpeosNodeRoot1 As OPTSpeosNodeRoot
   Set oPTSpeosNodeRoot1 = product1.GetItem("OPTVBExtension").SpeosCAA()
   Dim oPTSpeosNodeSimulation1 As OPTSpeosNodeSimulation
   Set oPTSpeosNodeSimulation1 = oPTSpeosNodeRoot1.Item("Simulations")
   Set oPTSimuDirect1 = oPTSpeosNodeSimulation1.Item("Lightguide")

6. We proceed the same way to create a reference to the 3D texture and have it updated in order to make screen shots. A references to the “Optical Properties” node is used:
   Dim oPTSpeosNodeOpticalProperties1 As OPTSpeosNodeOpticalProperties
   Set oPTSpeosNodeOpticalProperties1 = oPTSpeosNodeRoot1.Item("Optical Properties")
   Set oPTSpeosFeature1 = oPTSpeosNodeOpticalProperties1.Item("Lightguide")

7. The initialization is finished with the following lines:
   ' Initialisation
   FolderPath = productDocument1.Path
Set XMPViewer = CreateObject("xmpviewer.application")
Set fso = CreateObject("scripting.filesystemobject")

If Not fso.FolderExists(FolderPath & "\Results") Then
    fso.CreateFolder (FolderPath & "\Results")
End If

Call WriteInfo("Initialization check.")

XMPfile = productDocument1.Path & "\SPEOS output files\Backlight_unit.Lightguide.Irradiance sensor.1.xmp"
AreaListCount = 7
simuNumber = 1

- FolderPath corresponds to the path of the project’s directory.
- fso is a File System Object that will be used to write a report and to call the FolderExists and CreateFolder functions.
- XMPfile is a reference to the XMP result generated by the direct simulation.
- AreaListCount corresponds to the number of areas the optimization is going to consider.
- simuNumber is the parameter linked to the number of simulation updates.

Lesson 3: Writing a Report

With the following function you can write a line in a report file every time it is called. The line is formatted in order to include the date and time the function is summoned. The File System Object fso checks is the report already exists in order to create or open it.

Sub WriteInfo(myline)

    myline = Format(Str(Now), "yyyy/MM/dd H:mm:ss") & " " & myline

    Set fso = CreateObject("scripting.filesystemobject")

    Dim ftextfile

    If Not fso.FileExists(FolderPath & "\Report.txt") Then
        Set ftextfile = fso.createtextfile(FolderPath & "\Report.txt", True)
        ftextfile.writeline myline
        ftextfile.Close
        Set ftextfile = Nothing
    Else
        Set ftextfile = fso.opentextfile(FolderPath & "\Report.txt", 8)
        ftextfile.writeline myline
        ftextfile.Close
        Set ftextfile = Nothing
    End If

End Sub

Lesson 4: Writing the Optimization Loop

1. Back to the CATMain routine, the following loop goes through each of the 8 areas and calls the optimization loop.

    Call WriteInfo("Optimization process.")
Dim PassNumber As Integer

For PassNumber = 0 To AreaListCount Step 1
    Call WriteInfo(PassNumber + 1 & " e pass.")
    Call OptimizationLoop(PassNumber)
    Call SaveFinalXMP(PassNumber)
Next PassNumber

2. The OptimizationLoop routine first initializes the parameters corresponding to the maximum, minimum and average dot spacing. The three simulation update flags are also set to false.

Sub OptimizationLoop(PassNumber As Integer)
    DotRep_Max = param_pitch_max.Value
    DotRep_Min = param_pitch_min.Value
    DotRep_Avg = (DotRep_Max + DotRep_Min) / 2
    SimuInf = False
    SimuMid = False
    SimuSup = False

Then the following Do loop calls 3 times the CalcDeltaE routine before the Minimize routine until reaching a certain difference between the maximum and the minimum dot spacing.

    Do
        Call CalcDeltaE(DotRep_Min, DeltaEInf, SimuInf, PassNumber)
        WriteInfo "Min " & DotRep_Min & " " & DeltaEInf
        Call CalcDeltaE(DotRep_Avg, DeltaEMid, SimuMid, PassNumber)
        WriteInfo "Avg " & DotRep_Avg & " " & DeltaEMid
        Call CalcDeltaE(DotRep_Max, DeltaESup, SimuSup, PassNumber)
        WriteInfo "Max " & DotRep_Max & " " & DeltaESup
        Call Minimize(DotRep_Min, DeltaEInf, SimuInf, DotRep_Avg, DeltaEMid, SimuMid, DotRep_Max, DeltaESup, SimuSup)
    Loop Until (Abs(DotRep_Max - DotRep_Min) < 0.001)

3. The CalcDeltaE routine checks if the simulation is up to date or not. In case the simulation is not up to date, the rest of the routine is executed.

Sub CalcDeltaE(DotRep As Double, DeltaE As Double, simu As Boolean, LineNumber As Integer)
    Dim Measure(7)
    If simu = False Then
    End If
End Sub

First, the pitch corresponding to the active area of the backlight takes the value of the dot spacing in order to activate the reaction and generate the new 3D texture mapping.

If LineNumber = 0 Then
    param_pitch1.Value = DotRep
ElseIf LineNumber = 1 Then
    param_pitch2.Value = DotRep
ElseIf LineNumber = 2 Then
    param_pitch3.Value = DotRep
ElseIf LineNumber = 3 Then
    param_pitch4.Value = DotRep
ElseIf LineNumber = 4 Then
  param_pitch5.Value = DotRep
ElseIf LineNumber = 5 Then
  param_pitch6.Value = DotRep
ElseIf LineNumber = 6 Then
  param_pitch7.Value = DotRep
ElseIf LineNumber = 7 Then
  param_pitch8.Value = DotRep
End If

product1.Update
oPTSpecsFeature1.Update

Then, the simulation is updated and the result is analyzed.
Call WriteInfo("SPEOS Core simulation."
Call WriteInfo("Simulation result analysis.")
Call XMPAnalysis(Measure)

Finally, the merit function is calculated and the simulation flag set to true.
DeltaE = Sqr((Measure(LineNumber) - targetflux.Value) * (Measure(LineNumber) - targetflux.Value))
simu = True

4. The algorithm used in the Minimize routine reduces the range between the minimum and the maximum dot spacing, making the average value converging to the best solution. The simulation status flags are modified consequently to avoid useless calculations.

Sub Minimize(XInf As Double, YInf As Double, UpToDateInf As Boolean, XMid As Double, YMid As Double, UpToDateMid As Boolean, XSup As Double, YSup As Double, UpToDateSup As Boolean)
  If (YMid < YInf) Then
    If (YMid < YSup) Then
      If (YInf < YSup) Then
        XSup = XMid: YSup = YMid
        XMid = (XInf + XSup) / 2: UpToDateMid = False
      Else
        XInf = XMid: YInf = YMid
        XMid = (XInf + XSup) / 2: UpToDateMid = False
      End If
    Else
      XInf = XMid: YInf = YMid
      XMid = XSup: YMid = YSup
      XSup = XMid + (XMid - XInf): UpToDateSup = False
    End If
  Else
    XInf = XMid: YInf = YMid
    XMid = XSup: YMid = YSup
    XSup = XMid + (XMid - XInf): UpToDateSup = False
  End If
Else
  If (YMid < YSup) Then
    XSup = XMid: YSup = YMid
    XMid = XInf: YMid = YSup
    XInf = XMid - (XSup - XMid): UpToDateInf = False
    If XInf < param_pitch_min.Value Then
      XInf = param_pitch_min.Value: UpToDateInf = True
      XMid = (XInf + XSup) / 2: UpToDateMid = False
    End If
  Else
    If (YInf < YSup) Then
      XSup = XMid: YSup = YMid
      XMid = XInf: YMid = YSup
      XInf = XMid - (XSup - XMid): UpToDateInf = False
      If XInf < param_pitch_min.Value Then
        XInf = param_pitch_min.Value: UpToDateInf = True
        XMid = (XInf + XSup) / 2: UpToDateMid = False
      End If
    Else
      XInf = XMid: YInf = YMid
      XMid = XSup: YMid = YSup
      XSup = XMid + (XMid - XInf): UpToDateSup = False
    End If
  End If
XMid = XInf: YMid = YInf
XInf = XMid - (XSup - XMid): UpToDateInf = False

If XInf < param_pitch_min.Value Then
  XInf = param_pitch_min.Value: UpToDateInf = True
  XMid = (XInf + XSup) / 2: UpToDateMid = False
End If
Else
  XInf = XMid: YInf = YMid
  XMid = XSup: YMid = YSup
  XSup = XMid + (XMid - XInf): UpToDateSup = False
End If
End If
End Sub

Lesson 5: Saving Optimization Result

The routine SaveFinalXMP opens the last XMP result of the optimization and saves a copy of it. It also writes the value of each area’s pitch in a text file.

Sub SaveFinalXMP(PassNumber)

Dim retval
retval = XMPViewer.OpenFile(XMPfile)
retval = XMPViewer.ExportXMP(FolderPath & "\Results\" & PassNumber + 1 & "ePasse.xmp", 0, 0, 0, 0)

Set fso = CreateObject("scripting.filesystemobject")

Dim ftextfile As Object
Set ftextfile = fso.createtextfile(FolderPath & "\Results\Spacing.txt", True)

ftextfile.writeline "distribution at " & PassNumber + 1 & "e Pass"
ftextfile.writeline param_pitch1.Value
ftextfile.writeline param_pitch2.Value
ftextfile.writeline param_pitch3.Value
ftextfile.writeline param_pitch4.Value
ftextfile.writeline param_pitch5.Value
ftextfile.writeline param_pitch6.Value
ftextfile.writeline param_pitch7.Value
ftextfile.writeline param_pitch8.Value
ftextfile.Close

Set ftextfile = Nothing

End Sub

Lesson 6: Analyzing Simulation Result

The routine XMPAnalysis opens the XMP result and measures the flux over each defined areas. A JPEG picture corresponding to the result is saved in the Results directory. The simulation number is incremented.

Sub XMPAnalysis(Measure)

Dim retval, dYPos, dYWidth

End Sub
retval = XMPViewer.OpenFile(XMPfile)

For i = 0 To AreaListCount - 1 Step 1
    dYPos = -XMPViewer.YHeight / 2 + (i + 1 / 2) * XMPViewer.YHeight / AreaListCount
    dYWidth = XMPViewer.YHeight / AreaListCount
    retval = XMPViewer.SurfaceRectangleCalculation(0, dYPos, XMPViewer.XWidth, dYWidth)
    Measure(i) = Format(retval(7), ".000e+00")
Next i

Dim SimuName
SimuName = FolderPath & "\Results" & Format(simuNumber, "0000") & ".jpg"
retval = XMPViewer.ExportXMPImage(SimuName, 3)
simuNumber = simuNumber + 1

End Sub

The code of the macro is here.

**Lesson 6: Launching the Optimization**

1. From CATIA's tree, right-click Reaction.1, Reaction.1 object, and then select Deactivate.
2. Expand the Parameters node.
3. Double-click param_pitch1 to param_pitch7, and then set their value to 0.033.
4. Right-click Reaction.1, Reaction.1 object, and then select Activate.
5. Double-click param_pitch8 and then set it value to 0.033.
6. From the specification tree, expand the Simulations node.
7. Select Lightguide, and then click Local Update (Update).
8. From the specification tree, double-click the .xmp file to open the result.

9. Click Surface / Section.

10. Check the value in the Flux box.
    The value should be around 20.74lm.

11. Close the result.

12. From CATIA's tree, double-click targetflux, right-click in the value box, and then select Edit formula.

13. Enter \( \frac{20.74 \text{lm}}{12} \), and then click OK.

14. Change back the value of the parameters param_pitch1 to param_pitch8 to 1.

15. Expand the Lightguide node, and then double-click Number of rays.

16. Right-click the value and then select Edit formula.

17. In the Members of All group box, double-click param_raynb, and then click OK.

18. Click Tools, Macro, Macro..., and then double-click Module1.

19. Press F5 to run the macro.

**Lesson 7: Analyzing Results**

1. Open the Report.txt file located in the root directory of the project.
    This file contains all the actions recording using the WriteInfo routine.
You can analyze the optimization results of the first area using a Stock chart presenting the evolution of the maximum, minimum and average spacing values.

The spacing range decreases with the number of iterations until reaching the final value.

The following table summarizes the results:

<table>
<thead>
<tr>
<th>Area</th>
<th>Area 2</th>
<th>Area 3</th>
<th>Area 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of simulation updates</td>
<td>20</td>
<td>19</td>
<td>19</td>
</tr>
<tr>
<td>Difference between measured and target flux (lm)</td>
<td>0.018</td>
<td>0.031</td>
<td>0.086</td>
</tr>
</tbody>
</table>

XMP result

<table>
<thead>
<tr>
<th>Area 5</th>
<th>Area 6</th>
<th>Area 7</th>
<th>Area 8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of simulation updates</td>
<td>21</td>
<td>18</td>
<td>22</td>
</tr>
<tr>
<td>Difference between measured and target flux (lm)</td>
<td>0.096</td>
<td>0.058</td>
<td>0.034</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0.303</td>
</tr>
</tbody>
</table>
Creating a Backlight Unit Optimization 2

You must have the S_SV5_LM2 package with the O_SV5_3DTEXT and O_SV5_OPTIM2 options.
You must have CATIA R19 version or higher.

Lesson 1: Opening the Project

1. Open SPEOS CAA V5 Based VXX.
2. Click Tools, Options..., General, Parameters and Measure to edit the Knowledge tab.
3. Select the With value and With formula check boxes.
4. From the Knowledge Environment tab, select the Load extended language libraries and All packages check boxes.
5. From the tree, expand the Infrastructure node, and then select Part Infrastructure.
6. From the Display tab, select the Constraints, Parameters and Relations check boxes.
7. Click OK.
9. From your recent created folder, open the Backlight_unit.CATProduct file.

Lesson 2: Creating Parameters

1. Click Tools, Formula....
2. Click New parameter of type.
3. Select Real type with Single Value.
4. Rename it param_x_min and set its value to -24.
5. Repeat these steps to create the following parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>param_x_max</td>
<td>Real</td>
<td>24</td>
</tr>
<tr>
<td>param_y_min</td>
<td>Real</td>
<td>0</td>
</tr>
<tr>
<td>param_y_max</td>
<td>Real</td>
<td>71</td>
</tr>
<tr>
<td>param_pitch_min</td>
<td>Real</td>
<td>0.033</td>
</tr>
<tr>
<td>param_pitch_max</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch1</td>
<td>Real</td>
<td>1</td>
</tr>
</tbody>
</table>
### Lesson 3: Creating Measures

1. From the specification tree, select Lightguide, and then click Local Update (Update).

2. Click Measure (Measure).

3. In the XMP box, select Backlight_unit.Lightguide.Irradiance sensor.1.xmp.

4. Select Surface from the list.

5. Click Preview.

6. From the Outputs list, select Flux.

7. Click OK.

8. Repeat steps 2 to 6 seven times.

   The eight measures appear in the specification tree, in the Measures section.

### Lesson 4: Creating Formulas

Formulas are used in order to parameterize.

- Each measurement previously defined to cover one height of the sensor' surface.
- Each parameter corresponding to a merit function.

1. From the specification tree, expand the Measures node, and then expand the Measure.1 node.

2. Double-click Center Y, right-click in the value box, and then select Edit formula....

3. Type -param_y_max*7mm/16.

4. Click OK.

5. Click OK to validate.

6. Repeat steps 2 to 5 for the Height and Width parameters.

7. Set the formula for the Height and Weight parameter of each measurement.

8. Set the formula for each Center Y parameter of the measurements as defined below:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Type</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>param_pitch2</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch3</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch4</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch5</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch6</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch7</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_pitch8</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit1</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit2</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit3</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit4</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit5</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit6</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit7</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>merit8</td>
<td>Real</td>
<td>1</td>
</tr>
<tr>
<td>param_raynb</td>
<td>Integer</td>
<td>10,000,000</td>
</tr>
<tr>
<td>targetflux</td>
<td>Luminous flux</td>
<td>0</td>
</tr>
</tbody>
</table>
### MEASURE | PARAMETER | FORMULA
--- | --- | ---
Measure.2 | Center Y | -param_y_max \* 5mm/16
Measure.3 | Center Y | -param_y_max \* 3mm/16
Measure.4 | Center Y | -param_y_max \* 1mm/16
Measure.5 | Center Y | param_y_max \* 1mm/16
Measure.6 | Center Y | param_y_max \* 3mm/16
Measure.7 | Center Y | param_y_max \* 5mm/16
Measure.8 | Center Y | param_y_max \* 7mm/16

9. From the specification tree, double click merit1.
10. Right-click in the value box, and then select Edit formula....
11. Type \( \sqrt{(targetflux - \text{flux})} \), and then click Flux under Measure.1 to add it to the formula.
12. Type **)**2 to finish the formula.
13. Click OK.

   The following warning appears:
   
   "Units are not homogeneous (cdsr into Constant). We advise you to specify precise units for constants, otherwise the International System Units will be the default. (Ex:10\*10mm or MyRealParameter\*MyRealParameter*1mm)"

14. Click OK.
15. Click OK.
16. Click OK.
17. Repeat steps 9 to 16 for merit2, merit3, merit4, merit5, merit6, merit7 and merit8 using the corresponding Measure.

   You must select the flux corresponding to the measure.

18. In the specification tree, expand the Lightguide node, and then double-click Number of rays.
19. Right-click in the value box, and then select Edit formula....
20. In the Members of All group box, click param_raynb.
21. Click OK.
22. Click OK to validate.

### Lesson 5: Creating a VB Script

1. Click Start, Knowledgeware, Knowledge Advisor, and then click Macros with arguments (Actions).
2. In the Definition group box, in the argument(s) box, type the name of all the created parameters, separating each other by a comma.
3. Type the VB code by following the steps.
   - Start the VB script by typing the following line to set the arguments.
     
     ```vb
     Sub main(param_x_min, param_x_max, param_y_min, param_y_max, param_pitch_min, param_pitch_max, param_pitch1, param_pitch2, param_pitch3, param_pitch4, param_pitch5, param_pitch6, param_pitch7, param_pitch8, param_raynb, targetflux)
     ```
   - Continue with the following lines to make a reference to the active document.
     ```vb
     Dim productDocument1 As Document
     Set productDocument1 = CATIA.ActiveDocument
     ```
     ```vb
     Dim product1 As Product
     Set product1 = productDocument1.Product
     ```
   - The following code changes the number of rays of the direct simulation in order to indicate its status is not up to date and therefore the simulation needs to be run.
param_raynb.Value = 1000
product1.Update

param_raynb.Value = 1000000
product1.Update

- Then, type a File System Object in order to create a temporary text file that includes the 3D texture distribution.

    Dim fso As Object
    Set fso = CreateObject("Scripting.FileSystemObject")

    Dim f As Object
    Set f = fso.CreateTextFile(CATIA.ActiveDocument.Path & "\HalfSphere_dist_temp.txt", True)

- Some variables have to be defined and initialized.
  - IsOdd represents the flag indicating whether a vertical line of the 3D texture distributed is odd or pair.
    Dim IsOdd As Boolean
    IsOdd = True
  - dot_spacingX and dot_spacingY indicate respectively the spacing along the X and Y directions.
  - dot_posX and dot_posY indicate respectively the dot position along the X and Y directions.
  - dot_numb indicates the number of dots.
  - NumbAreas represents the number of areas.
  - areaNumber represents the actual area number.

    Dim dot_spacingX, dot_spacingY, dot_posX, dot_posY, dot_numb, NumbAreas, areaNumber
    dot_spacingY = 0
dot_posY = 0
dot_numb = 0
NumbAreas = 8
areaNumber = 1
    dot_spacingX = param_pitch1.Value

- Do While loop increments the value of the dot_posY parameter until it reaches the maximum length defined by param_y_max.Value - param_pitch_min.Value.

    Do While dot_posY < (param_y_max.Value - param_pitch_min.Value)

- Then, a test is made on the dot_spacingY parameter as it should not be lower than the minimum pitch value.

    If dot_spacingY < param_pitch_min.Value Then
        dot_spacingY = param_pitch1.Value
    End If

- The IsOdd parameter is checked in order to initialize the dot position along the X direction. If the line is odd, it starts with the centred dot, with two symmetrical dots otherwise. The number of dots is incremented accordingly.
If IsOdd = True Then
    dot_posX = 0
    f.writeline dot_posX & " " & dot_posY & " 0 1 0 0 1 0 1 1 1"
    dot_numb = dot_numb + 1
Else
    dot_posX = dot_spacingX / 2
    f.writeline dot_posX & " " & dot_posY & " 0 1 0 0 1 0 1 1 1"
    f.writeline -dot_posX & " " & dot_posY & " 0 1 0 0 1 0 1 1 1"
    dot_numb = dot_numb + 2
End If

- The line is continued until the dot position along the X direction reaches a maximum length value.

    Do While Abs(dot_posX) <= (param_x_max.Value - param_pitch_min.Value)
        dot_posX = dot_posX + dot_spacingX
        f.writeline dot_posX & " " & dot_posY & " 0 1 0 0 1 0 1 1 1"
        f.writeline -dot_posX & " " & dot_posY & " 0 1 0 0 1 0 1 1 1"
        dot_numb = dot_numb + 2
    Loop

- The dot_posY parameter is checked to see if the dot is still located in the corresponding area. If it is not the case, the area number is incremented.

    If dot_posY >= areastart And dot_posY < areaend Then
        'nothing
    Else
        areaNumber = areaNumber + 1
    End If

- Then, the dot_posY parameter takes the value of the corresponding pitch according to the area number.

    If areaNumber = 1 Then
        dot_spacingY = param_pitch1.Value
    ElseIf areaNumber = 2 Then
        dot_spacingY = param_pitch2.Value
    ElseIf areaNumber = 3 Then
        dot_spacingY = param_pitch3.Value
    ElseIf areaNumber = 4 Then
        dot_spacingY = param_pitch4.Value
    ElseIf areaNumber = 5 Then
        dot_spacingY = param_pitch5.Value
    ElseIf areaNumber = 6 Then
        dot_spacingY = param_pitch6.Value
    ElseIf areaNumber = 7 Then
        dot_spacingY = param_pitch7.Value
    ElseIf areaNumber = 8 Then
        dot_spacingY = param_pitch8.Value
    End If

- The dot position along the Y direction is then incremented. The dot_spacingX parameter is then modified according to the new value of the pitch along Y direction.

    dot_posY = dot_posY + dot_spacingY
    dot_spacingX = 2 * dot_spacingY / Sqr(3)

- Finally, the IsOdd flag is updated.

    If IsOdd = True Then
        IsOdd = False
    Else
        IsOdd = True
    End If

This concludes the “Do While” loop.

- The text file is closed. It is then open to be read.
f.Close
Set f = fso.OpenTextFile(CATIA.ActiveDocument.Path & "\HalfSphere_dist_temp.txt", 1)

- A new text file is created and corresponds this time to the actual 3D mapping file. Its first line corresponds to
  the number of dots.

Dim ftemp As Object
Set ftemp = fso.CreateTextFile(CATIA.ActiveDocument.Path & "\SPEOS input
files\BLU_mapping.OPT3DMapping", True)
ftemp.writeline dot_numb

- A loop then writes the content of the first text file in the second one. Both text files are then closed.

Do While f.AtEndOfStream <> True
    ftemp.writeline f.ReadLine
Loop
ftemp.Close
Set ftemp = Nothing
f.Close
Set f = Nothing
End Sub

1. Click OK to validate the script.
   The script appears in the CATIA’s tree, in the Relations section.
   The code of the macro is here.

Lesson 6: Creating a Reaction

1. Click Start, Knowledgeware, Knowledge Advisor, and then click Reactions (Reactive Features).
2. In the Sources box, select param_pitch1 to param_pitch8.
   param_pitch1 to param_pitch8 parameters have to be in the same order than in the VB script.
3. Click Edit action....
   The Action Editor: Reaction dialog box appears.
4. Double click VB Script.1 in the tree.
5. From the Dictionary group box, select Messages and macros.
6. From the Members of Messages and macros group box, double-click VB Script -> Run.
7. In the Inputs box, select all the created parameters.
   Parameters have to be in the same order than in the VB script
8. Click OK.
9. Click OK to validate the Reaction.
   The reaction appears in the CATIA’s tree.
10. Test the reaction by changing the value of param_pitch1 for example.
    The result of the reaction is the .txt file creation.

Lesson 7: Creating Optimizations

1. Click Start, Knowledgeware, Product Engineering Optimizer.
2. Click Optimization (Product Engineering Optimizer).
3. In the Optimized parameter group box, click Select....
4. From CATIA’s tree click merit1, and then click OK.
5. In the Free Parameters group box, click Edit list.
6. From the Parameters list, double-click param_pitch1.
7. Click OK.
8. In the Free Parameters group box, select param_pitch1, and then click Edit ranges and step.
9. In the Inf. Range check box type 0.033.
10. In the Sup. Range check box type 1.
11. In the Step check box type 0.1.
12. Click OK.
13. From the Available algorithms group box, select Gradient Algorithm Without Constraint from the list.
14. In the Running Criteria, from the Convergence speed list, select Medium.
15. In the Termination criteria group box, in the Maximum number of updates box, type 200.
16. In the Consecutive updates without improvements box, type 50.
17. Click to clear the Maximum time (minutes) check box, and then type 10000 in the box.
18. Check that the Save optimization data check box is selected.
19. Click OK to validate the optimization creation.
20. Repeat these steps using merit2 to merit8 parameters as Optimized parameter and param_pitch2 to param_pitch8 as Free Parameters.

Lesson 8: Launching the Optimization

1. From CATIA's tree, right-click Reaction.1, Reaction.1 object, and then select Deactivate.
2. Expand the Parameters node.
3. Double-click param_pitch1 to param_pitch7, and then set their value to 0.033.
4. Right-click Reaction.1, Reaction.1 object, and then select Activate.
5. Double-click param_pitch8 and then set it value to 0.033.
6. From the specification tree, expand the Simulations node.
7. Select Lightguide, and then click Local Update (Update).
8. From the specification tree, double-click the .xmp file to open the result.
9. Click Surface / Section.
10. Check the value in the Flux box.
   The value should be around 20.74 lm.
11. Close the result.
12. From CATIA’s tree, double-click targetflux, right-click in the value box, and then select Edit formula.
13. Enter 20.74 lm/12, and then click OK.
14. Change back the value of the parameters param_pitch1 to param_pitch8 to 1 using the same method.
15. Expand the Lightguide node, and then double-click Number of rays.
16. Right-click the value and then select Edit formula.
17. In the Members of All group box, double-click param_raynb, and then click OK.
18. Double-click Optimization.1.
19. Click Run optimization.
20. Enter the filename of optimization report, and then click Save.
21. Once the process is finished, repeat these steps for each of the height optimizations.

Lesson 9: Analyzing Results
1. From CATIA’s tree, double-click Optimization.1.
2. From the Computations results tab, click Show curves....
   The following dialog box appears, presenting the Optimized parameter, Free Parameters and best value variation as a function of the iteration number.

3. Repeat these steps for each of the eight optimizations.
   The following table summarizes the results:

<table>
<thead>
<tr>
<th></th>
<th>Area 1</th>
<th>Area 2</th>
<th>Area 3</th>
<th>Area 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of simulation updates</td>
<td>25</td>
<td>24</td>
<td>22</td>
<td>21</td>
</tr>
<tr>
<td>Difference between measured and target flux (lm)</td>
<td>0.0024</td>
<td>0.00018</td>
<td>0.0062</td>
<td>9.6x10^-6</td>
</tr>
</tbody>
</table>
Setting Polarizer Surface

You must have the S_SV5_LM1 or S_SV5_LM2 or S_SV5_LM4 solution with the O_SV5_POL option.

With this tutorial, you are about to learn the introduction of polarization management in SPEOS for CATIA, applied for electronics applications.

You are about to create an anti-glaring filter on mobile screen composed by a linear polarizer and a quarter-wave plate.

20 minutes
Lesson 1: Preparing Data

2. Open SPEOS CAA V5 Based VXX.
3. Open the POLA_Mobile Phone.CATProduct file.

Lesson 2: Creating a Polarizer Surface

1. Click Polarization plate.
2. In geometry, select the parameters of the Axis System group box as defined below:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Origin</td>
<td>Vertex/ARF_POLARIZER_O</td>
</tr>
<tr>
<td>X Direction</td>
<td>Edge/Axis_X</td>
</tr>
<tr>
<td>Y Direction</td>
<td>Edge/AXIS_Y</td>
</tr>
</tbody>
</table>

These coordinates define the orientation of the polarization plate axes.

3. Select both Mirrored Extent boxes of the Axis System group box.
4. Set the parameters of the Plate Geometry group box according to the following definition:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>X Start</td>
<td>-27.8mm</td>
</tr>
<tr>
<td>X End</td>
<td>27.8mm</td>
</tr>
<tr>
<td>Y Start</td>
<td>-39.10mm</td>
</tr>
<tr>
<td>Y End</td>
<td>39.10mm</td>
</tr>
</tbody>
</table>

These parameters define the size of the polarization plate.

5. In the Polarization group box, select Library in Type list.
6. Click to open the Polarizer Surface Editor.
7. Write in the description box your comment, like:
   Linear polarizer - created by Author, Date
   
   For more details, you can view Polarizer Surface Editor.
8. Set all reals part of Jones matrix to 1 and all imaginary part to 0, and close the Polarizer Surface Editor.
9. Save your polarization file, in SPEOS input files folder of tutorial.
   You can save it as LinearPolarizer.polarizer
10. In the File box, browse the LinearPolarizer.polarizer files.
11. Click OK.
12. From Specification tree, right click on Polarization Plate.1 feature.
13. Click Properties.
14. Rename feature as Polarizer in feature name box.
15. Click OK.
   Polarizer is set.
   You can create now the quarter-wave plate.
16. Click Polarization plate.
   The property manager appears.
17. Select, in geometry, parameters of Axis system group box as defined below:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Origin</td>
<td>Vertex/ARF_QUARTER-WAVE_O</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>X Direction</td>
<td>Edge/Axis_X</td>
</tr>
<tr>
<td>Y Direction</td>
<td>Edge/AXIS_Y</td>
</tr>
</tbody>
</table>

Make sure ARF_POLARIZER_O and ARF_QUARTER-WAVE_O are really close. Zoom in 3D view to differentiate both points.

This geometrical coordinate defines the orientation of quarter-wave plate axis.

18. Check both Mirrored Extent box of Plate geometry group box.
19. Set parameters of Plate Geometry group box according to the following definition:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>X max</td>
<td>27.8mm</td>
</tr>
<tr>
<td>Y max</td>
<td>39.10mm</td>
</tr>
</tbody>
</table>

These parameters define size of the quarter-wave plate.

20. In the Polarization group box, select Quarter wave plate in Type list.
21. Click OK.
22. From Specification tree, right click on Polarization Plate.2 feature.
23. Click Properties.
24. Rename feature as Quarter-wave in feature name box.
25. Click OK.

Your quarter-wave plate is now set.

Lesson 3: Analyzing Anti-glaring Filter with Ray Tracing

1. From the specification tree, expand the Simulations node, and then right-click the RAY TRAC interactive simulation.
2. Click Hide/Show.
3. Select the RAY TRAC interactive simulation, and then click Local Update (Update).

Interactive simulation shows light passing through front glass of the mobile phone and its reflection on an internal component.

4. From the specification tree, double-click RAY TRAC interactive simulation.
5. Select the Geometries box.
6. Select polarization plate Polarizer and Quarter-wave.
7. Click OK.
8. Select the RAY TRAC interactive simulation, and then click Local Update (Update).
10. Right-click, and then select Hide/Show.

With this section, you can see light passing through front glass and reflecting on upper surface of display. This time, light reflected is stopped on Polarizer.

Light comes across front glass and goes through the polarizer. Output light is linearly polarized.
Light goes through the quarter-wave plate and becomes circular polarized. Then part of this light is reflected on an optical interface. The polarization phase is shifted by 180°.
Then the reflected light hits the quarter wave plate and is linearly polarized with its orientation crossed with the polarizer. Reflected light is finally stopped by the polarizer.

Creating an Infrared Detection System

You must have the S_SV5_LM4 solution.

The main goal of this tutorial is to show the most important parameters to set to create a Short-Wave Infrared detection system.
In this tutorial the distance between thermic sources and the detection system is 150 meters.
1 Hour + 1 Hour for the simulation.

Lesson 1: Opening the Project

You must have the S_SV5_LM4 solution.

2. Open SPEOS CAA V5 Based VX.
3. From your recently created folder, open SV5_Tutorial_DetectionSystem_R19V14\DATA\Pdt_Tutorial_DetectionSystem.CATProduct file.

Lesson 2: Creating Optical Properties

You must have the S_SV5_LM4 solution.

1. Click Simple Scattering Surface Editor (Editors).
2. Set the parameters as follows:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Sand</td>
</tr>
<tr>
<td>Absorption</td>
<td>95</td>
</tr>
<tr>
<td>Reflection</td>
<td>Tick</td>
</tr>
</tbody>
</table>
3. Click Save the active document.
4. Browse the file:
   `SV5_Tutorial_DetectionSystem_R19V14\DATA\Scene\SPEOS input files`
5. Change the document name as Sand and click Save.
6. Close the Scattering surface dialog box.
   Sand.simplescattering is now created.
7. On CATIA menu, click File, Open...
8. Browse `SV5_Tutorial_DetectionSystem_R19V14\DATA\ImagingSystem\ImagingSystem.CATMaterial`.
9. Click Open.
10. In Optic tab, right-click N-LASF31A, and click Copy.
11. Paste this copy in Optic tab.
12. Right-click Copy_of_N-LASF31A, and then select Properties.
13. In the Properties dialog box, click More...
14. Select Optical Properties tab. In Surface optical properties (SOP) group box, choose Library type.
15. Browse AR_coating.simplescattering file. Click Open.
16. Select the Feature Properties tab, in the Feature Name box type AR-N-LASF31A. Then click OK.
   The AR-N-LASF31A, with anti-reflective coating is created.
17. In Optic tab, right-click N-SF57, then Copy.
18. Paste this copy in Optic tab.
19. Right-click Copy_of_N-SF57, and then select Properties.
20. In the Properties dialog box, click More...
21. Select Optical Properties tab. In Surface optical properties (SOP) group box, choose Library type.
22. Browse AR_coating.simplescattering file. Click Open.
23. Select the Feature Properties tab, in the Feature Name box type AR-N-SF57. Then click OK.

The AR-N-SF57, with anti-reflective coating is created.
It is important to let open ImagingSystem.CATMaterial dialog box.
24. Return to the CATProduct.
25. Click Apply Material (Optical Properties).
26. In Library dialog box, browse Default Material Catalog in current material library.
27. Select Metal tab.
28. Right-click on Brushed metal 2, and select Copy.
29. Select ImagingSystem.CATMaterial window box, paste the material in Other tab.
30. Right-click on this material (Brushed Metal 2), and then select Properties.
31. In the Properties dialog box, click More...

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reflection</td>
<td></td>
</tr>
<tr>
<td>Gaussian angle</td>
<td>25</td>
</tr>
<tr>
<td>Lambertian</td>
<td>100</td>
</tr>
<tr>
<td>Gaussian</td>
<td>0</td>
</tr>
</tbody>
</table>
32. If the Warning dialog box appears, click OK.
33. In Volume optical properties (VOP) group box, choose Opaque Type.
34. In Surface optical properties (SOP) group box, choose Library Type.
35. Browse Metal_anodized.anisotropicbsdf file. Click Open.
36. Select Feature Properties tab, in Feature Name box type Anodized-Metal. Then OK.
   The Anodized-Metal is now created in ImagingSystem.CATMaterial Library.
37. Save ImagingSystem.CATMaterial.
38. Close the CATMaterial file.

Lesson 3: Applying Optical Properties

You must have the S_SV5_LM4 solution.

1. Double-click on Pdt_ImagingSystem in CATIA's trees, to define it as work object.
2. From CATIA's tree, expand Pdt_ImagingSystem node and expand the Prt_OpticalSystem node.
3. Select Lens1 body.
4. Click Apply Material (Optical Properties).
5. Click Open a material library.
6. Browse the file SV5_Tutorial_DetectionSystem_R19V14\DATA\ImagingSystem\ImagingSystem.CATMaterial.
7. From the Library dialog box, in the Optic tab, select the AR-N-LAS31A material and click OK.
   The material appears in Lens1 body section.
8. Apply the following materials to other bodies of Prt_OpticalSystem and Prt_Imager.

<table>
<thead>
<tr>
<th>Body</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lens2</td>
<td>AR-N-SF57</td>
</tr>
<tr>
<td>Lens3</td>
<td>AR-N-LASF31A</td>
</tr>
<tr>
<td>Mounting</td>
<td>Anodized-Metal</td>
</tr>
<tr>
<td>Clips</td>
<td>Anodized-Metal</td>
</tr>
<tr>
<td>CCD</td>
<td>CCD</td>
</tr>
</tbody>
</table>

Materials are now set in OpticalSystem's bodies.

9. Right-click on Prt_OpticalSystem. Then click Hide/Show to hide the part.
10. Expand Prt_Imager node. Right-click on CCD body and click Reframe on.
11. Click Face Optical Properties (Optical Properties).
12. Select the front face of CCD, in the graphics area.
13. Select Library Type on Surface optical properties (SOP) group box.
14. Browse InGaAs-responsivity.coated. Then click OK.
   Note that this coated corresponds to Indium Gallium Arsenide coating. This type of material is commonly used on Short-Wave Infrared Sensors.
15. Right-click on Face Optical Properties.1, created above, and select Properties.
16. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type InGaAs_coating.
   InGaAs coating is now created.
17. Show all components.
Lesson 4: Creating Thermic Source

You must have the S_SV5_LM4 solution.

Creation of Tank Thermic source
1. Double-click Pdt_LandSystem in CATIA's tree, to define it as work object.
2. Click Thermic Surface Source (Sources).
3. Expand Pdt_Scene node, expand Pdt_LandSystem node. Then expand AxisSystem geometrical set node.
4. In the Thermic Source Definition dialog box, select Radiant unit in Flux group box.
5. In Emittance group box, select Temperature field type.
6. Select Origin in Axis system group box. From CATIA's tree, select Tank point.
7. Select X direction in Axis system group box. From CATIA's tree, select Y line.
8. Select Y direction in Axis system group box. From CATIA's tree, select Z line.
9. Select Temperature field group box.
10. Browse SV5_Tutorial_DetectionSystem_R19V14\DATA\Scene\LandSystem\SPEOS input files\LandSystem.OPTTemperatureField.
11. Click Open.
12. Select Library type in the Surface optical properties group box.
13. Browse the file SV5_Tutorial_DetectionSystem_R19V14\DATA\Scene\LandSystem\SPEOS input files\LandSystem.scattering.
14. Click Open. Then click OK on Thermic Source Definition dialog box.
15. Right-click the Thermic source.1 created above and select Properties.
16. In Properties dialog box, select Feature Properties tab, in the Feature Name box, type Tank_Thermic_Source. Then click OK.
   The Tank thermic source is now created.

Creation of Soldier Thermic Source
1. Double-click Pdt_Soldier(Pdt_Soldier.1) in CATIA's tree, to define it as work object.
2. Right-click on Pdt_Soldier, and select Reframe on.
3. Click Thermic Surface Source (Sources).
4. In Thermic Source Definition dialog box, select Radiant unit in the Flux group box.
5. In Emittance group box select Emissive faces type. Set the value to 310 in the Temperature spinbox.
6. In the Emissive faces box, click all the Skin and Clothes faces in the graphics area. Then click OK.

7. Right-click on Thermic source.1. Then select Properties.
8. In the Properties dialog box, select the Feature Properties tab.
9. In the Feature Name box, type Soldier_Thermic_Source and click OK.
   The Soldier thermic source is now created.

Note that the second and the third soldiers are linked to the first one. So the three thermic sources are created.

Creation of Ground Thermic Source.
1. Double-click Pdt_Scene in CATIA's tree, to define it as work object.
2. Expand Pdt_Scene node, expand Prt_Ground node. Then expand Ground geometrical set node and expand Ground sweep node.
3. Select Sand and click Edit SPEOS properties (Tools).
4. In Surface Optical Properties (SOP) group box, select Library type.
5. Select Sand.simplescattering created on Lesson 2 and click OK.
   The scattering is now applied on the ground.
6. Click Thermic Surface Source (Sources).
7. In Thermic Source Definition dialog box, select Radiant Unit in Flux group box.
8. In Emittance group box select Emissive faces type. Set the Temperature spinbox to 300.
9. In the Emissive faces box, click on the Ground front face in the graphics area. Then click OK.
10. Right-click on Thermic source.1, select Properties.
11. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type Ground_Thermic_Source. Then click OK.
   The Ground thermic source is now created.

Lesson 5: Creating an Irradiance SWIR Sensor

You must have the S_SV5_LM4 solution.
1. Double-click Pdt_ImagingSystem in CATIA's tree, to define it as work object.
2. Expand the Prt_Imager node, and expand References geometrical set node.
3. Right-click on References and select High/Show to show it.

4. Click Irradiance Sensor (Sensor).

5. In Irradiance Sensor Definition dialog box, set the parameters as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Colorimetric</td>
</tr>
<tr>
<td>Ray File</td>
<td>False</td>
</tr>
<tr>
<td>Origin</td>
<td>Origin (References geometrical set)</td>
</tr>
<tr>
<td>X Direction</td>
<td>X (References geometrical set)</td>
</tr>
<tr>
<td>Y Direction</td>
<td>Y (References geometrical set)</td>
</tr>
<tr>
<td>Integration Type</td>
<td>Planar</td>
</tr>
<tr>
<td>Integration Direction</td>
<td>No selection</td>
</tr>
<tr>
<td>X End</td>
<td>4.8</td>
</tr>
<tr>
<td>X Sampling</td>
<td>320</td>
</tr>
<tr>
<td>Y End</td>
<td>3.84</td>
</tr>
<tr>
<td>Sampling</td>
<td>256</td>
</tr>
<tr>
<td>Both X &amp; Y Mirror Extent</td>
<td>Tick</td>
</tr>
<tr>
<td>Wavelength Start</td>
<td>900</td>
</tr>
<tr>
<td>Wavelength End</td>
<td>1650</td>
</tr>
<tr>
<td>Sampling</td>
<td>7</td>
</tr>
</tbody>
</table>

6. Click More...

7. In Layer box, select Source.

8. In CATIA's tree, expand the Prt_OpticalSystem node.

9. Right-click on the Housing body of the Prt_OpticalSystem. Then select Hide/Show to hide the body.

10. In Output faces for inverse simulation optimization box, select the nearest face of the Lens3 body in the graphics area.

11. Click Ok.

12. Right-click on Irradiance sensor.1, select Properties. Click More...


14. In Step group box, set the value of the X spinbox to 0.5.

15. Set the value of the Y spinbox to 0.5.

16. Select Feature Properties tab, in Feature Name box, type SWIR_Sensor. Click OK.

The Short-Wave Infrared Sensor is now created.

**Lesson 6: Creating an Interactive Simulation for FOV analysis**

*You must have the S_SV5_LM4 solution.*

**Creating Sources**

1. Double-click Pdt_ImagingSystem in CATIA's tree, to define it as work object.

2. Expand Prt_OpticalSystem node. Expand the Analysis geometrical set node. Then right-click on Analysis geometrical set and select Hide/Show, to show it.
3. Click on Interactive Source (Sources).
4. In Interactive Source Definition dialog box, select Curve-Direction in Type box.
5. Set the value of the Wavelength spinbox to 1275.
   1275nm corresponds to the central wavelength of the sensor spectral band.
6. In Start group box, select the line Size of 0.00deg geometrical set in Curve box. Set the Sampling box value to 20.
7. In End group box, select the line Axis of 0.00deg geometrical set in Direction box.
8. Click Reverse direction. Then click OK.
9. Right-click on Interactive source.1, select Properties.
10. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type Source_0.00deg. Then click OK.
11. Right-click on Source_0.00deg, select Copy.
12. Right-click, then select Paste.
13. Double-click on Source_0.00deg.1
14. In Interactive Source Definition dialog box, select the line Size of 2.15deg geometrical set in Curve box.
15. In End group box, select the line Axis of 2.15deg geometrical set in Direction box. Then click OK.
16. Right-click on Source_0.00deg.1, select Properties.
17. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type Source_2.15deg. Then click OK.
18. Right-click on Source_0.00deg, select Copy.
19. Right-click, then select Paste.
20. Double-click on Source_0.00deg.1.
21. In Interactive Source Definition dialog box, select the line Size of -2.15deg geometrical set in Curve box.
22. In End group box, select the line Axis of -2.15deg geometrical set in Direction box. Then click OK.
23. Right-click on Source_0.00deg.1, select Properties.
24. In the Properties dialog box, select Feature Properties tab, in the Feature Name box, type Source_-2.15deg. Then click OK.
   All sources are now created.

Creating the Interactive Simulation

1. Click Interactive Simulation (Simulations).
2. In the Sources box, select Source_0.00deg.
3. In the Geometries box, select Lens1, Lens2, Lens3, Mounting, Clips, CCD.
4. In the Sensor box, select SWIR_Sensor. Then click OK.
5. Right-click Interactive simulation.1, select Properties.
6. Click More… Then select Simulation tab.
7. Set the value of the Geometrical distance tolerance spinbox to 0.025.
8. In Weight box, select False.
9. Select Feature Properties tab, in the Feature Name box, type Simulation_0.00deg. Then click OK.
10. Right-click on Simulation_0.00deg, select Copy.
11. Right-click, then select Paste.
12. Double-click on Simulation_0.00deg.1.
13. In the Source box, select Source_2.15deg. Then click OK.
14. Right-click on Simulation_0.00deg.1, select Properties.
15. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type Simulation.2.15deg. Then click OK.
16. Right-click on Simulation_0.00deg, select Copy.
17. Right-click, then select Paste.

18. Double-click on Simulation_0.00deg.1.

19. In the Source box, select Source_-2.15deg. Then click OK.

20. Right-click on Simulation_0.00deg.1, select Properties.

21. In the Properties dialog box, select the Feature Properties tab, and in the Feature Name box, type Simulation-2.15deg. Then click OK.

   The Simulation is now created.

   It is used to determine the FOV of the system. You can change the SWIR_Sensor length and verify the FOV analysis by changing the angle between different axes in Analysis Geometrical set.

![Simulation Diagram]

Note that the ray color has been changed in the picture above. Default ray color is the true color linked to wavelength. In the case of IR wavelength, the associated color is black.

To change the color, right-click on the simulation in CATIA's tree, then click Properties. On the Graphic tab, select a color in the Graphic Properties group box.

---

**Lesson 7: Creating an Inverse Simulation**

*You must have the S_SV5_LM4 solution.*

1. Double-click Pdt_Demo_DetectionSystem in CATIA's tree, to define it as work object.

2. Click Inverse Simulation (Simulations).

3. In the Sources box, select Ground_Thermic_Source, Tank_Thermic_Source, Soldier_Thermic_Source, Soldier_Thermic_Source, Soldier_Thermic_Source.

   They correspond to the 3 soldiers’ thermic sources.

4. In the Geometries box, select Lens1, Lens2, Lens3, Mounting, Clips, CCD.

5. In the Sensor box, select SWIR_Sensor.

6. Set the value of the Number of pass spinbox to 80. Click OK.

7. Right-click on Inverse simulation.1, select Properties.

8. In Properties dialog box, click More... Then select Simulation tab.

9. Set the value of the Tessellation step value spinbox to 10.

10. Set the value of the Geometrical distance tolerance spinbox to 0.025.

11. In Weight box, select True.

12. Set the value of the Minimum energy percentage spinbox to 0.5.

13. Select Inverse simulation tab.


15. Select Feature Properties tab.

16. In Feature Name box, type Target_Simulation. Then click OK.

   The simulation of the target is now created.

---

**Lesson 8: Running & Analyzing an Inverse Simulation**

*You must have the S_SV5_LM4 solution.*
1. From the specification tree, select Target_Simulation, then click Local Update (Update). The Optical Properties Error window appears.

2. Click Ignore, to run the simulation without the dispersion phenomena. The simulation lasts approximatively an hour on an Intel® Xeon® E5649 2,53Ghz (2 processors).

3. From the specification tree, expand the Target_Simulation node.

4. Double-click the .xmp file.

5. Click View and select Radiometric units.

6. Change the color scale from True Color to Black to White (color). The result of the simulation is not in visible scale because sources used are thermic ones.

7. Click Level .

8. Change the scale from Linear to Log.

9. Click on the lowest value.

10. Type 2e-9 in Value box.

The image is inside out because of the detection system design.

Creating a Rear Position Lamp with Light Guides

You must have CATIA R18 or more with:
S_SV5_OSD (Lesson 2)
S_SV5_LM2 (Lesson 3)
S_SV5_VE3 (Lesson 4)
O_SV5_3DED (Lesson 3)
O_SV5_VR (Lesson 4)
O_SV5_COL (Lesson 4)
O_SV5_OPTIM1 (Lesson 4)

2 hours (plus simulation time)
You are going to create a rear position lamp with light guides features.

Lesson 1: Opening Project


2. Launch SPEOS CAA V5 Based.

Lesson 2: Designing Light Guide Geometries with Optical Shape Design (OSD)

Lesson 2.1: Creating Light Guides

You must have the S_SV5_OSD option.

1. Double-click the Rear Position Lamp Light Guides part from CATIA’s tree.
2. Right-click on the LightGuides body from the Rear Position Lamp Light Guides part and select Define In Work Object.
   - The light guide is created in a body because it is a solid (not a surface).
3. Click Start, Mechanical Design, Part Design to change the workbench.
   - You need to be in this workbench to create OSD solids and make basic operations on them.
4. Click Light Guide (Optical Shape Design).
   - The Light Guide Definition dialog box appears.
   - Feel free to temporarily hide some features in order to better see the geometrical inputs.
   - To do so, select the features to hide, and click Hide/Show.
5. Set the following parameters:
   1. Body tab

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Guide Curve</td>
<td>Rear Position Lamp Light Guides/LightGuides/Guides/Guide.1</td>
</tr>
<tr>
<td>Body/Type</td>
<td>Constant Profile</td>
</tr>
<tr>
<td>Body/Profile</td>
<td>Rear Position Lamp Light Guides/LightGuides/Profiles/Profile.1</td>
</tr>
<tr>
<td>Prisms/Type</td>
<td>Direction</td>
</tr>
<tr>
<td>Prisms/Optical Axis</td>
<td>Rear Position Lamp Light Guides/AxisSystem/OpticalAxis</td>
</tr>
</tbody>
</table>

   2. Prisms tab

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shape</td>
<td>Flat</td>
</tr>
<tr>
<td>Distances/Mode</td>
<td>Curvilinear</td>
</tr>
<tr>
<td>Distances/Start</td>
<td>10mm</td>
</tr>
<tr>
<td>Distances/End</td>
<td>0mm</td>
</tr>
<tr>
<td>Distances/Step</td>
<td>Constant</td>
</tr>
<tr>
<td>Distances/Step/Value</td>
<td>2mm</td>
</tr>
<tr>
<td>Distances/Length</td>
<td>Automatic</td>
</tr>
<tr>
<td>Offset</td>
<td>Constant</td>
</tr>
<tr>
<td>Offset/Value</td>
<td>5mm</td>
</tr>
<tr>
<td>Width</td>
<td>Constant</td>
</tr>
<tr>
<td>Width/Value</td>
<td>6mm</td>
</tr>
<tr>
<td>Start Angle</td>
<td>Design Table</td>
</tr>
<tr>
<td>End Angle</td>
<td>Design Table</td>
</tr>
<tr>
<td>Orientation</td>
<td>Normal</td>
</tr>
</tbody>
</table>

   3. Manufacturing tab

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Milling</td>
<td>False</td>
</tr>
</tbody>
</table>
Note that the design table status changes to Not defined and that a warning icon appears next to the Start Angle and End Angle parameters. This means that Start Angle and End Angle are driven by a design table which has not yet been defined. You are going to load a design table where the definition of these parameters has ever been done using Excel's formulas.

6. Click Create new / Use existing....
7. Load the LightGuide.xlsx file.
   Notice that the design table status changes to OK (green traffic light icon), which means that its definition is complete. Note that Optical Axis and Light Direction (depicted in yellow arrows) are not correctly oriented in the 3D view.

8. From the Body tab, click Reverse Direction for the Guide Curve.
9. From the Body tab, click Reverse Direction for the Optical Axis.
   You notice in the 3D view that Optical Axis and Light Direction (depicted in yellow arrows) are now correctly oriented.

10. Click Preview.
11. Click OK.
   The feature is created and appears in the 3D view and on the specification tree.

12. Rename the feature into Top Light Guide.
13. Repeat the steps 4 to 11 with the following parameters to create the middle light guide:

   1. Body tab

      | PARAMETER     | DEFINITION                                                                 |
      |---------------|-----------------------------------------------------------------------------|
      | Guide Curve   | Rear Position Lamp Light Guides/LightGuides/Guides/Guide.2                   |
      | Body/Type     | Constant Profile                                                            |
      | Body/Profile  | Rear Position Lamp Light Guides/LightGuides/Profiles/Profile.2               |
      | Prisms/Type   | Direction                                                                   |
      | Prisms/Optical Axis | Rear Position Lamp Light Guides/AxisSystem/OpticalAxis                     |

   2. Prisms tab

      | PARAMETER | DEFINITION |
      |-----------|------------|
      | Shape     | Flat       |
3. Manufacturing tab

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Milling</td>
<td>False</td>
</tr>
<tr>
<td>Milling/Value</td>
<td>0.2mm</td>
</tr>
</tbody>
</table>

The feature is created and appears in the 3D view and on the specification tree.

14. Rename the feature into Middle Light Guide.

15. Repeat the steps 4 to 11 with the following parameters to create the bottom light guide:

1. **Body tab**

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Guide Curve</td>
<td>Rear Position Lamp Light Guides/LightGuides/Guides/Guide.3</td>
</tr>
<tr>
<td>Body/Type</td>
<td>Constant Profile</td>
</tr>
<tr>
<td>Body/Profile</td>
<td>Rear Position Lamp Light Guides/LightGuides/Profiles/Profile.3</td>
</tr>
<tr>
<td>Prisms/Type</td>
<td>Direction</td>
</tr>
<tr>
<td>Prisms/Optical Axis</td>
<td>Rear Position Lamp Light Guides/AxisSystem/OpticalAxis</td>
</tr>
</tbody>
</table>

2. **Prisms tab**

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shape</td>
<td>Flat</td>
</tr>
<tr>
<td>Distances/Mode</td>
<td>Curvilinear</td>
</tr>
<tr>
<td>Distances/Start</td>
<td>10mm</td>
</tr>
<tr>
<td>Distances/End</td>
<td>0mm</td>
</tr>
<tr>
<td>Distances/Step</td>
<td>Constant</td>
</tr>
</tbody>
</table>
### Lesson 2.2: Applying Material

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA’s tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.

4. Click Apply Material (Optical Properties).
   The Library dialog box opens.
5. Choose Default Material Catalog.
6. In the Other tab, select the Plexiglass material.
7. Click OK.
   The material appears in the 3D view and in the specification tree, just under the LightGuides body.

You can create a catalog of materials containing all the materials that you are regularly using, in order not to have to define them each time.

Both volume optical properties (VOP) and surface optical properties (VOP) can be taken into account for the materials of this catalog.
Lesson 3: Validating Photometrical Performances with Light Modeling (LM)

Lesson 3.1: Adding Optical Properties
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click on the Plexiglass material applied to the LightGuides body inside the Rear Position Lamp Light Guides part.
4. Click Edit SPEOS properties (Tools).
   If a warning saying New applicative properties have been added and will be saved with the current material appears, click OK.
5. In the Volume optical properties (VOP) group box:
   - Select Library from the Type list.
   - Click ... and load the Cyro_Acrylic_7N.material file from the SPEOS input files folder.
6. In the Surface optical properties (SOP) group box, select Optical polished from the Type list.
7. Click Close.

Lesson 3.2: Adding Face Optical Properties
You are going to add some face optical properties (FOP) on the smooth curved surface of each light guides in order to simulate diffusing effect.
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Face Optical Properties (Optical Properties).
4. In the Faces group, select the smooth curved surface of each light guides from the 3D view.
5. In the Surface optical properties (SOP) group box:
   - Select Library from the Type list.
   - Click ... and load the Charmille n°33.unpolished file from the SPEOS input files folder.
   Note that this file, and more generally any other surface files, can be downloaded from the OPTIS portal.
6. Click OK.
7. Rename the face optical properties feature into Charmille.
8. Hide the face optical property.

Lesson 3.3: Creating Sensors

Lesson 3.3.1: Creating Intensity Sensor
You are going to create an intensity sensor in order to analyze the intensity distribution of the rear position lamp.
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Intensity Sensor (Sensors).
4. Set the following parameters:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>PARAMETER</td>
<td>DEFINITION</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Type</td>
<td>Photometric</td>
</tr>
<tr>
<td>Format</td>
<td>XMP</td>
</tr>
<tr>
<td>Near field</td>
<td>False</td>
</tr>
<tr>
<td>Axis System/Origin</td>
<td>Rear Position Lamp Light Guides/Sensors/Intensity/Origin</td>
</tr>
<tr>
<td>Axis System/X Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/Intensity/X</td>
</tr>
<tr>
<td>Axis System/Y Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/Intensity/Y</td>
</tr>
<tr>
<td>X/Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>X/End</td>
<td>62.5deg</td>
</tr>
<tr>
<td>X/Sampling</td>
<td>500</td>
</tr>
<tr>
<td>Y/Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>Y/End</td>
<td>15deg</td>
</tr>
<tr>
<td>Y/Sampling</td>
<td>120</td>
</tr>
<tr>
<td>Orientation</td>
<td>X As Meridian, Y As Parallel</td>
</tr>
</tbody>
</table>

5. Click OK.
   The sensor appears in the specification tree and in the 3D view.

6. Rename the sensor into Intensity.
   You can see the grid of the intensity sensor on the display, fixed by X Step and Y Step parameters.
   This grid is automatically used by the result files generated by the simulations using this sensor.

7. Right-click on the intensity sensor.
8. Click Properties.
9. Click More....
10. In the Grid tab, set respectively X Step and Y Step to 10deg and 5deg.
11. Click OK.
    The grid is adjusted in the 3D view.
12. Hide the sensor.

Lesson 3.3.2: Creating 3D Irradiance Sensor
You are going to create a 3D irradiance sensor in order to analyze the energy distribution on the emissive surfaces of the light guides.
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click 3D Irradiance Sensor (Sensors).
4. Select as Faces the three smooth curved surfaces of the light guides.

The light is expected to be extracted by the prisms and pass through these three surfaces.

5. Click OK.

The sensor appears in the specification tree and in the 3D view.

6. Rename the sensor into 3D Irradiance.

7. Hide the sensor.

**Lesson 3.3.3: Creating 3D Energy Density Sensor**

*You must have the O_SV5_3DED option.*

You are going to create a 3D energy density sensor in order to analyze the energy distribution inside the light guides.

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA’s tree.

2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.

3. Click 3D Energy Density Sensor (Sensors).

4. Set the following parameters:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Photometric</td>
</tr>
<tr>
<td>Axis System/Origin</td>
<td>Rear Position Lamp Light Guides/Sensors/3D Energy Density/Origin</td>
</tr>
<tr>
<td>Axis System/X Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/3D Energy Density/X</td>
</tr>
<tr>
<td>Axis System/Y Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/3D Energy Density/Y (*)</td>
</tr>
<tr>
<td>Dimensions/X Size</td>
<td>300mm</td>
</tr>
<tr>
<td>Dimensions/Y Size</td>
<td>80mm</td>
</tr>
<tr>
<td>Dimensions/Z Size</td>
<td>50mm</td>
</tr>
<tr>
<td>Sampling/X Sampling</td>
<td>600</td>
</tr>
<tr>
<td>Sampling/Y Sampling</td>
<td>160</td>
</tr>
<tr>
<td>Sampling/Z Sampling</td>
<td>100</td>
</tr>
</tbody>
</table>

(*) Y Direction cannot be selected from the specification tree but has to be selected from the 3D view (limitation on rotate features).

5. Click OK.
The sensor appears in the specification tree and in the 3D view.

6. Rename the sensor into 3D Energy Density.
7. Hide the sensor.

**Lesson 3.4: Creating Simulations**

**Lesson 3.4.1: Creating Direct Simulation with LM**

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Direct Simulation (Simulations).
4. In the Sources box, select the six LED_OSRAM_LS_E63F_RAYS sources respectively from the six LED_OSRAM_LS_E63F products.
   These sources have been created using a ray file coming from OSRAM website. For more details about ray file source, you can view Ray File Source (see page 49).
5. In the Geometries box, select the following features:
   - LightGuides body from the Rear Position Lamp Light Guides part.
   - Outer.Lens body from the Rear Position Lamp Light Guides part.
   - Bezel split feature on the Bezel geometrical set from the Rear Position Lamp Light Guides part.
6. In the Sensors box, select both Intensity and 3D Irradiance sensors from the specification tree.
7. Type 1e7 for Number of rays.
8. In the Geometries box, click on the LightGuides feature.
   The tessellation of the light guides is displayed.
   Note that the meshes are too large according to the size of a prism.
10. Click OK.
    The direct simulation appears in the specification tree.
11. Rename the simulation into LM.
12. Right-click on LM.
13. Click Properties.
14. Click More....
15. Go to the Simulation tab.
16. Set Tessellation sag value to 200.
17. Set Tessellation step value to 10.
18. Click OK.
19. Open the direct simulation and check the tessellation of the light guides again by repeating steps 8 and 9. We notice that the meshes are fine enough according to the size of the prism now.

20. Right-click on LM.
22. Click More....
23. Go to the Simulation tab.
24. Set Smart Engine to 12.
   - This option allows you to reduce the simulation time by a better management of the computer memory. For more details, you can view Smart Engine (see page 141).
25. Set 0.001mm as Geometrical distance tolerance.
   - This setting allows you to avoid volume body not closed error due to the high number of edges of the light guide. For more details about volume body not closed error, you can view Understanding Propagation Errors (see page 148).
26. Click OK.

Lesson 3.4.2: Creating Direct Simulation with 3DED
You must have the O_SV5_3DED option.

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. Copy/paste the LM direct simulation.
3. Rename the pasted simulation into 3DED.
4. Edit the 3DED simulation.
5. In the Geometries box, unselect all the features except LightGuides.
6. In the Sensors box, unselect all the sensors and select 3D Energy Density sensor.
7. Click OK.

Lesson 3.5: Running Simulations
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Select the LM and 3DED simulations from the specification tree.
4. Click Local Update (Update).
   - The simulations are running.
   - The simulation lasts approximately 15 minutes on an Intel® Xeon® E5620 2,40Ghz (2 processors). When the simulations are complete, the results appear in the 3D view.
5. Hide the simulations.

Lesson 3.6: Isolating Results
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Select the LM and 3DED simulations from the specification tree.
Note that the simulations are regarded as up to date features according to their icons.

4. Click Isolate (Simulations).
   The isolated simulations appear at the bottom of the Simulations node of the specification tree.
   They are physically stored inside the SPEOS isolated files folder.
5. Rename the isolated simulations into LM and 3DED.
6. Hide the isolated simulations from the specification tree.

**Lesson 3.7: Analyzing Results**

**Lesson 3.7.1: Analyzing Intensity Result**

1. In the specification tree, double-click on the XMP result of the LM isolated simulation.
   The result opens in the Virtual Photometric Lab.
   Note that the steps of the grid are retrieved from the sensor.
2. Right-click on the map.
3. Select Show ruler.
4. Click Level.
5. Tick IsoCurve.
6. Click Initial Size.

7. Click Save.
9. You want to check that the ECE R7 regulation for rear position lamp is passed.
11. Reach the Library section and select Standards Type.
   This section gathers some HTML files for standards validation.
12. Click on Photometry - ECE R7.
13. Click Download, I agree and Save as.
   The file is downloading.
14. Open the Motor vehicles - ECE R7 Regulation Rev.5 - Rear position lamp.html file from this folder.
15. Click Browse....
16. Select the XMP file of the LM isolated XMP file in the SPEOS isolated files folder.
17. Click Process.
Green and/or red fills should appear on the cells saying if the test points are passed or failed.

<table>
<thead>
<tr>
<th>Test points</th>
<th>Min (cd)</th>
<th>Value (cd)</th>
<th>Max (cd)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10U</td>
<td>0.8</td>
<td>5.28</td>
<td>17</td>
</tr>
<tr>
<td>20L</td>
<td>0.4</td>
<td>5.01</td>
<td>17</td>
</tr>
<tr>
<td>10L</td>
<td>0.8</td>
<td>7.53</td>
<td>17</td>
</tr>
<tr>
<td>5L</td>
<td>2.8</td>
<td>6.20</td>
<td>17</td>
</tr>
<tr>
<td>10R</td>
<td>0.8</td>
<td>4.81</td>
<td>17</td>
</tr>
<tr>
<td>20R</td>
<td>0.4</td>
<td>3.07</td>
<td>17</td>
</tr>
<tr>
<td>10D</td>
<td>1.4</td>
<td>5.42</td>
<td>17</td>
</tr>
<tr>
<td>3D</td>
<td>3.6</td>
<td>7.19</td>
<td>17</td>
</tr>
<tr>
<td>V</td>
<td>4.0</td>
<td>6.57</td>
<td>17</td>
</tr>
<tr>
<td>V</td>
<td>3.6</td>
<td>6.07</td>
<td>17</td>
</tr>
<tr>
<td>10R</td>
<td>1.4</td>
<td>5.28</td>
<td>17</td>
</tr>
<tr>
<td>20R</td>
<td>0.4</td>
<td>4.80</td>
<td>17</td>
</tr>
<tr>
<td>10R</td>
<td>0.8</td>
<td>4.84</td>
<td>17</td>
</tr>
<tr>
<td>20R</td>
<td>0.4</td>
<td>2.60</td>
<td>17</td>
</tr>
<tr>
<td>10D</td>
<td>0.8</td>
<td>5.05</td>
<td>17</td>
</tr>
<tr>
<td>3D</td>
<td>0.8</td>
<td>4.31</td>
<td>17</td>
</tr>
<tr>
<td>S</td>
<td>0.6</td>
<td>0.26</td>
<td>...</td>
</tr>
</tbody>
</table>

You have passed all the test points here so the regulation is passed.

Lesson 3.7.2: Analyzing 3D Irradiance Result

1. In the specification tree, double-click on the XM3 result of the LM isolated simulation.
   The result opens in the Virtual 3D Photometric Lab.
   - You want to check that the light is well extracted by the prisms, so you are interested in the transmission.
   2. Select Transmission (Photometric units).
   - You need to adapt the illuminance scale to better see the variations on the output surface of the light guide.
   3. Set Max to 1200 lux.

Areas with highest values of illuminance are areas where the most light is extracted by the prisms.
You can notice hot spots on the left of the light guides due to light injection by the LEDs.

4. Click Save.
5. Close the Virtual 3D Photometric Lab.

Lesson 3.7.3: Analyzing 3D Energy Density Result

1. In the specification tree, double-click on the VMP result of the 3DED isolated simulation.
   The result opens in the 3D Energy Density Lab.
   - Some dark areas appear outside of the light guide body but we want to focus on what happens inside.
   2. Click Level.
   3. Set the minimum value of the photometric scale to 3000 lm/m3.
   - The energy density is displayed just inside the light guide body using a transparency effect.
   4. Set the maximum value of the photometric scale to 200000 lm/m3.
The energy density inside the light guide body is now easier to get.

5. Tick Cutting views from the Properties field.
   You can better see the energy density inside the light guide body when using cutting views.
   You can also change the cutting planes to check the energy density at other places of the light guide body.

6. Tick Decorations from the Properties field.

7. Click 

8. Make a translation of a cutting view by holding a click on the red, green or blue cube and moving the mouse at the same time.
   You can also change the X, Y and Z values from the Position field of the Cutting manipulator Editor.

9. Make a rotation of a cutting view by holding a click on the red, green or blue cube and moving the mouse at the same time, while also holding down Ctrl.
   You can also change the X, Y and Z values from the Orientation field of the Cutting manipulator Editor.

10. Click Save and erase the existing file.

11. Close the 3D Energy Density Lab.

Lesson 4: Assessing Lit Appearance with Visual Ergonomics (VE)

Lesson 4.1: Creating an Ambient Source

1. Log on OPTIS website (http://www.optis-world.com).
2. Reach the Library section and select Environment Type.
   This section gathers some links toward websites offering HDR files.
3. Click on Openfootage HDR.
4. Click Download, I agree and Open.
   The openfootage.net website appears.
5. Find HDRI 360° Czech republic and download the file with the best resolution (130MB).
6. Unzip the archive and move the HDR file in the SPEOS input files folder.
7. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
8. If the Visual Ergonomics workbench is not launched yet, click Start, Ergonomics Design & Analysis, Visual Ergonomics.
9. Click Ambient Source (Sources).
10. Select the following parameters:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Environment</td>
</tr>
<tr>
<td>Zenith</td>
<td>Rear Position Lamp Light Guides/AxisSystem/Z</td>
</tr>
<tr>
<td>Properties/North direction</td>
<td>Rear Position Lamp Light Guides/AxisSystem/OpticalAxis</td>
</tr>
<tr>
<td>Properties/Luminance</td>
<td>15000cd/m²</td>
</tr>
<tr>
<td>Properties/Red spectrum</td>
<td>\SPEOS input files\Gaussian red.spectrum</td>
</tr>
<tr>
<td>Properties/Green spectrum</td>
<td>\SPEOS input files\Gaussian green.spectrum</td>
</tr>
<tr>
<td>Properties/Blue spectrum</td>
<td>\SPEOS input files\Gaussian blue.spectrum</td>
</tr>
<tr>
<td>Properties/Environment type</td>
<td>HDRI File</td>
</tr>
<tr>
<td>Properties/HDRI file</td>
<td>\SPEOS input files\OpenfootageNET_Czech republic_High.hdr</td>
</tr>
<tr>
<td>Ground plane/Origin</td>
<td>Rear Position Lamp Light Guides/Ambient Source/Origin</td>
</tr>
<tr>
<td>Ground plane/Height</td>
<td>745mm</td>
</tr>
</tbody>
</table>

11. Click Preview.

12. Click Reverse direction for North direction.

13. Click Preview.

14. Click OK.

15. Rename the source into Environment.

16. Hide the source.

Lesson 4.2: Creating Sensors

Lesson 4.2.1: Creating Luminance Sensor

You must have the O_SV5.COL and O_SV5_OPTIM1 options.

You are going to create luminance sensors in order to make some renderings of the rear position lamp. The first one will be fixed in space and the second one will be moved in space using a design table to get an observer view.

Creating Luminance Sensor

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. Click Start, Ergonomics Design & Analysis, Visual Ergonomics if the workbench is not yet launched.
3. Click Radiance Sensor (Sensors).
4. Set the following parameters:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Colorimetric</td>
</tr>
</tbody>
</table>
PARAMETER | DEFINITION
---|---
Definition from | Point Line and Dimension
Axis System/Origin | Rear Position Lamp Light Guides/Sensors/Luminance/Origin
Axis System/X Direction | Rear Position Lamp Light Guides/Sensors/Luminance/X
Axis System/Y Direction | Rear Position Lamp Light Guides/Sensors/Luminance/Y
X/Mirrored Extent | True
X/End | 37mm
X/Sampling | 1680
Y/Mirrored Extent | True
Y/End | 12mm
Y/Sampling | 544
Wavelength/Start | 400nm
Wavelength/End | 700nm
Wavelength/Sampling | 13
Observer/Observer Type | Focal
Observer/Focal | 250mm

5. Tick Automatic framing.
   The 3D view is adjusted to see what the luminance result will look like.

6. Click OK.
   The sensor appears in the specification tree and in the 3D view.

7. Rename the sensor into Luminance.
8. Right-click on the sensor.
9. Click Properties.
10. Click More....
11. In the Parameters tab, set Integration angle to 2deg.
12. In the Parameters tab, set Save spectral data to true.
13. Click OK.
14. Hide the sensor.

Creating Luminance DT Sensor
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. Click Start, Analysis & Simulation, Light Modeling if the workbench is not yet launched.
3. Click Radiance Sensor (Sensors).
4. Set the following parameters:
   PARAMETER | DEFINITION
<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Colorimetric</td>
</tr>
<tr>
<td>Definition from</td>
<td>Point Line and Dimension</td>
</tr>
<tr>
<td>Axis System/Origin</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer DT/Origin</td>
</tr>
<tr>
<td>Axis System/X Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer DT/X</td>
</tr>
<tr>
<td>Axis System/Y Direction</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer DT/Y</td>
</tr>
<tr>
<td>X/Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>X/End</td>
<td>55mm</td>
</tr>
<tr>
<td>X/Sampling</td>
<td>1680</td>
</tr>
<tr>
<td>Y/Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>Y/End</td>
<td>20mm</td>
</tr>
<tr>
<td>Y/Sampling</td>
<td>615</td>
</tr>
<tr>
<td>Wavelength/Start</td>
<td>400nm</td>
</tr>
<tr>
<td>Wavelength/End</td>
<td>700nm</td>
</tr>
<tr>
<td>Wavelength/Sampling</td>
<td>13</td>
</tr>
<tr>
<td>Observer/Observer Type</td>
<td>Focal</td>
</tr>
<tr>
<td>Observer/Focal</td>
<td>250mm</td>
</tr>
<tr>
<td>Design Table</td>
<td>Relations/RotationHorizontaleVerticale</td>
</tr>
</tbody>
</table>

5. Click Reverse direction for X Direction.
6. Tick Automatic framing.
   The 3D view is adjusted to see what the luminance result looks like.

7. Click OK.
   The sensor appears in the specification tree and in the 3D view.

8. Rename the sensor into Luminance DT.
9. Open the RotationHorizontaleVerticale design table of the main product.
   Each line matches to a different configuration, which represents a different position for the sensor.
   Each position of the sensor is driven by two angles theta and phi being defined in the design table.
   The "<>" characters inform you that first configuration is active right now.
10. Double-click on another line to activate another configuration.
11. Click Update All.
    The position of the sensor has changed on the 3D view.
    The design table has been set as input of the sensor so all the configurations will be automatically dealt with during the simulation.
12. Right-click on the sensor.
13. Click Properties.
14. Click More....
15. In the Parameters tab, set Integration angle to 1deg.
16. In the Parameters tab, set Save spectral data to true.
17. Click OK.
18. Hide the sensor.

Lesson 4.2.2: Creating Observer Sensor

You must have the O_SV5_COL and the O_SV5_VR options.

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Visual Ergonomics workbench is not launched yet, click Start, Ergonomics Design & Analysis, Visual Ergonomics.
3. Click Observer Sensor (Sensors).
4. Set the following parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axis System/Origin</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer/Origin</td>
</tr>
<tr>
<td>Axis System/Horizontal</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer/Horizontal</td>
</tr>
<tr>
<td>Axis System/Vertical</td>
<td>Rear Position Lamp Light Guides/Sensors/Observer/Vertical</td>
</tr>
<tr>
<td>Wavelength/Start</td>
<td>400nm</td>
</tr>
<tr>
<td>Wavelength/End</td>
<td>700nm</td>
</tr>
<tr>
<td>Wavelength/Sampling</td>
<td>13</td>
</tr>
<tr>
<td>Vision Field/H Start</td>
<td>-110deg</td>
</tr>
<tr>
<td>Vision Field/H End</td>
<td>40deg</td>
</tr>
<tr>
<td>Vision Field/H Sampling</td>
<td>3</td>
</tr>
<tr>
<td>Vision Field/H Mirrored Extent</td>
<td>False</td>
</tr>
<tr>
<td>Vision Field/V Start</td>
<td>0deg</td>
</tr>
<tr>
<td>Vision Field/V End</td>
<td>10deg</td>
</tr>
<tr>
<td>Vision Field/V Sampling</td>
<td>3</td>
</tr>
<tr>
<td>Vision Field/V Mirrored Extent</td>
<td>False</td>
</tr>
<tr>
<td>Size/H Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>Size/H End</td>
<td>55mm</td>
</tr>
<tr>
<td>Size/H Sampling</td>
<td>1680</td>
</tr>
<tr>
<td>Size/V Mirrored Extent</td>
<td>True</td>
</tr>
<tr>
<td>Size/V End</td>
<td>20mm</td>
</tr>
<tr>
<td>Size/V Sampling</td>
<td>615</td>
</tr>
<tr>
<td>Distance</td>
<td>750mm</td>
</tr>
<tr>
<td>Focal</td>
<td>250mm</td>
</tr>
<tr>
<td>Stereo</td>
<td>False</td>
</tr>
<tr>
<td>Layer</td>
<td>Source</td>
</tr>
</tbody>
</table>

5. Tick Automatic framing.
The 3D view is adjusted to see how each point of view of the result is going to look like.

6. Set another values for H and/or V to change the point of view.

   ![3D view](image)

   This is useful to be sure that the size of the sensor is appropriate to the geometry. Basically we want to avoid the geometry to get out of the field of view for each point of view.

7. Click OK.

   The sensor appears in the specification tree and in the 3D view.

8. Rename the sensor into Observer.
9. Hide the sensor.

### Lesson 4.3: Creating Simulations

#### Lesson 4.3.1: Creating Luminance Simulations

You are going to create some direct and inverse simulations in order to characterize the rear position lamp. The simulation is done in 2 steps because direct simulations are required for light guide.

**Creating VE_LEDs Simulation**

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA’s tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Direct Simulation (Simulations).
4. In the Sources box, click the LM simulation.
   
   All the features defined as Sources for the LM simulation are retrieved for this new simulation.
5. In the Geometries box, click the LM simulation.
   
   All the features defined as Geometries for the LM simulation are retrieved for this new simulation.
6. In the Sensors box, select the Luminance sensor from the specification tree.
7. Type 2e8 for Number of rays.
8. Click OK.
9. Rename the simulation into VE_LEDs.
10. Right-click VE_LEDs.
11. Click Properties.
12. Click More....
13. Go to the Simulation tab.
14. Set Tessellation sag value to 200.
15. Set Tessellation step value to 10.
16. Set 0.001mm as Geometrical distance tolerance.
This setting allows you to avoid volume body not closed error due to the high number of edges of the light guide. For more details about volume body not closed error, you can view Understanding Propagation Errors (see page 148).

17. Set Smart Engine to 12.
   This option allows you to reduce the simulation time by a better management of the computer memory. For more details, you can view Smart Engine (see page 141).

18. Click OK.

**Creating VE_AmbientSource Simulation**

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Inverse Simulation (Simulations).
4. In the Sources box, click the Environment ambient source.
5. In the Geometries box, click the LM simulation.
6. All the features defined as Geometries for the LM simulation are retrieved for this new simulation.
7. In the Sensors box, select the Luminance sensor from the specification tree.
8. Type 100 for Number of pass.
9. Click OK.
10. Rename the simulation into VE_AmbientSource.
11. Right-click on VE_AmbientSource.
12. Click Properties.
13. Click More….
14. Go to the Simulation tab.
15. Set Tessellation sag value to 200.
16. Set Tessellation step value to 10.
17. Set 0.001mm as Geometrical distance tolerance.
   This setting allows you to avoid volume body not closed error due to the high number of edges of the light guide. For more details about volume body not closed error, you can view Understanding Propagation Errors (see page 148).
18. Go to the Inverse Simulation tab.
19. Set Splitting to true.
   This option provides a faster noise reduction on scenes with optical polished surface as the first surface state seen from the observer sensor. For more details, you can view the Splitting section of Using the Monte-Carlo Algorithm (see page 134).
20. Click OK.

**Lesson 4.3.2: Creating Observer Simulations**

You are going to create some direct and inverse simulations in order to characterize the rear position lamp. The simulation is done in 2 steps because direct simulations are required for light guide.

**Creating VR_LEDs Simulation**

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Click Direct Simulation (Simulations).
4. In the Sources box, click the LM simulation.
All the features defined as Sources for the LM simulation are retrieved for this new simulation.

5. In the Geometries box, click the LM simulation.

All the features defined as Geometries for the LM simulation are retrieved for this new simulation.

6. In the Sensors box, select the Luminance DT sensor from the specification tree.

7. Type 1e9 for Number of rays.

8. Click OK.

9. Rename the simulation into VR_LEDs.

10. Right-click VR_LEDs.

11. Click Properties.

12. Click More….

13. Go to the Simulation tab.

14. Set Tessellation sag value to 200.

15. Set Tessellation step value to 10.

16. Set 0.001mm as Geometrical distance tolerance.

This setting allows you to avoid volume body not closed error due to the high number of edges of the light guide.

For more details about volume body not closed error, you can view Understanding Propagation Errors (see page 148).

17. Set Smart Engine to 12.

This option allows you to reduce the simulation time by a better management of the computer memory.

For more details, you can view Smart Engine (see page 141).

18. Click OK.

Creating VR_AmbientSource Simulation

1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.

2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.

3. Click Inverse Simulation (Simulations).

4. In the Sources box, click the Environment ambient source.

5. In the Geometries box, click the LM simulation.

All the features defined as Geometries for the LM simulation are retrieved for this new simulation.

6. In the Sensors box, select the Observer sensor from the specification tree.

7. Type 500 for Number of pass.

8. Click OK.

9. Rename the simulation into VR_AmbientSource.

10. Right-click VR_AmbientSource.

11. Click Properties.

12. Click More….

13. Go to the Simulation tab.

14. Set Tessellation sag value to 200.

15. Set Tessellation step value to 10.

16. Set 0.001mm as Geometrical distance tolerance.

This setting allows you to avoid volume body not closed error due to the high number of edges of the light guide.

For more details about volume body not closed error, you can view Understanding Propagation Errors (see page 148).

17. Set Smart Engine to 12.

This option allows you to reduce the simulation time by a better management of the computer memory.

For more details, you can view Smart Engine (see page 141).
18. Go to the Inverse Simulation tab.
19. Set Splitting to true.
   This option provides a faster noise reduction on scenes with optical polished surface as the first surface state seen from the observer sensor.
   For more details, you can view the Splitting section of Using the Monte-Carlo Algorithm (see page 134).
20. Click OK.

**Lesson 4.4: Running Simulations**

**Lesson 4.4.1: Running Luminance Simulations**
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Select both VE_Leds and VE_AmbientSource simulations from the specification tree.
4. Click Local Update (Update).
   The simulations are running.
5. When the simulations are done, the results appear in the 3D view.
6. Hide the simulations.

**Lesson 4.4.2: Running Observer Simulations**
1. Double-click the Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Select both VR_Leds and VR_AmbientSource simulations from the specification tree.
4. Click Network Update (Update).
   The simulations are automatically isolated in the specification tree and exported.
   Two .sv5 files are created in the SPEOS isolated files folder, without the simulation's results.
   If an Optical Properties error message appears, click Ignore.
5. Open the Simulation Spooler Status application (Tools) to check and manage your simulation progress.
   The simulation lasts approximately 100 minutes on an Intel® Xeon® E5620 2,40Ghz (2 processors).
   The simulation lasts approximately 1 week on a cluster of 6 machines having each an Intel® Bi-Xeon® E5-2650 2Ghz (2 processors).
   Note that you can also open the simulation spooler status application from the start menu, by clicking All Programs, OPTIS, Distributed Computing, Simulation Spoolers Status.
6. Hide the simulations.

**Lesson 4.5: Isolating Results**
1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Light Modeling workbench is not launched yet, click Start, Analysis & Simulation, Light Modeling.
3. Select the VE_Leds and VE_AmbientSource simulations from the specification tree.
   Note that the simulations are regarded as up to date features according to their icons.
4. Click Isolate (Simulations).
   The isolated simulations appear at the bottom of the Simulations node of the specification tree.
   The isolated simulations are physically stored inside the SPEOS isolated files folder.
5. Rename the isolated simulations into VE_Leds and VE_AmbientSource.
6. Hide the isolated simulations from the specification tree.
   You can move up or down the isolated simulations from the specification tree using Alt+Ctrl+Up or Alt+Ctrl+Down.
Lesson 4.6: Combining Results

Lesson 4.6.1: Combining Luminance Results

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Visual Ergonomics workbench is not launched yet, click Start, Ergonomics Design & Analysis, Visual Ergonomics.
3. Click Photometric Calc (Tools).
   The Photometric Calc opens.
4. Click to browse the XMP results of the VE_LEDs and VE_AmbientSource isolated simulations as Source files.
5. Select Map union (combine two maps in one) as Operation.
6. Browse the destination of the final XMP file as Result.
7. Click Process.
   The XMP file of the final rendering is created.
8. Rename the file into VE_Final.XMP.

Lesson 4.6.2: Combining Observer Results

Creating VR_LEDs Result

1. Double-click Tutorials > Rear Position Lamp Light Guides product from CATIA's tree.
2. If the Visual Ergonomics workbench is not launched yet, click Start, Ergonomics Design & Analysis, Visual Ergonomics.
3. Click Virtual Reality Lab (Viewers).
   The Virtual Reality Lab opens.
4. Click Create an Observer View
5. Click Add.
6. Select all the XMP results of the VR_LEDs isolated simulation.
7. Click Create.
8. Select the destination for the final OptisVR file.
9. Click Save.
   The operation is processing.
10. Rename the file into VR_LEDs.OptisVR.

Creating VR_Final Result

1. Click Operations with OptisVR Files
2. Click Add.
3. Select the OptisVR file coming from the VR_AmbientSource isolated simulation and the VR_LEDs.OptisVR file obtained just before.
4. Select Union (combine layers) as Operation.
5. Click Create.
6. Select the destination for the final OptisVR file.
7. Click Save.
   The OptisVR file of the final observer view is created.
8. Rename the file into VR_Final.OptisVR.
Lesson 4.7: Analyzing Results

Lesson 4.7.1: Analyzing Luminance Result

1. Double-click the result VE_Final.XMP created before. The result opens in the Virtual Photometric Lab.

![Image of luminance result]

2. Click Initial Size. You can see the influence of the sources on the final rendering by activating/deactivating them individually.

3. Click Virtual lighting controller. You can see that all the sources appear (the environment and the six LEDs).

4. Untick the second and the fifth sources which match with the LEDs for the top light guide. The top light guide is not lit anymore.

![Image of luminance result with sources activated/deactivated]

To get a noise-free rendering, you have to run another row of simulations after having changed the value of some parameters.

5. Change the following parameters and run the concerned simulations again:

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>DEFINITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sensor/Luminance</td>
<td></td>
</tr>
<tr>
<td>Integration angle</td>
<td>1deg</td>
</tr>
<tr>
<td>Simulation/VE_LEDs</td>
<td></td>
</tr>
<tr>
<td>Number of rays</td>
<td>6e+009</td>
</tr>
<tr>
<td>Simulation/VE_AmbientSource</td>
<td></td>
</tr>
<tr>
<td>Number of pass</td>
<td>3000</td>
</tr>
</tbody>
</table>

The simulation lasts a week-end on an Intel® Xeon® E5620 2,40Ghz (2 processors).
After you having isolate and combine the results, the rendering is finally noise-free.

Lesson 4.7.2: Analyzing Observer Result
1. Double-click the VR_Final.OptisVR result created before.
   The result opens in the Virtual Reality Lab.
2. You can change the observer's point of view by moving the mouse while holding down left click and moving the mouse at the same time.
   You can also use the arrows key to change the observer's point of view.
INDEX

3

3D Energy Density Sensor • 93, 103
3D Irradiance Sensor • 92, 103
3D Texture • 20

A

Ambient Material • 20, 105, 108, 110
Ambient Source Visualization • 48
Ambient Source with CIE Standard General Sky Type • 42
Ambient Source with CIE Standard Overcast Sky Type • 41
Ambient Source with Environment Type • 44
Ambient Source with Natural Light Type • 43
Ambient Source with Uniform Type • 40
Ambient Sources • 40, 104
Applying VOP on Surface • 172
Automatic Mappings • 25
Axis System Definition • 21

B

Basics of Colorimetry • 12
Bump • 145

C

Changing Background Exposures • 49
Changing the User Interface Language • 11
Checking the Simulation Progress • 157
Checking the SPEOS CAA V5 Based License • 5
Circular Mapping • 25, 26
Colorimetry Limitations • 13
Creating a 3D Energy Density Sensor • 93
Creating a 3D Texture • 21
Creating a Ambient Source with CIE Standard General Sky Type • 42
Creating a Backlight Unit Optimization • 188
Creating a Backlight Unit Optimization 2 • 201
Creating a Direct Simulation • 107
Creating a LCD Component • 31
Creating a Light Modeling System • 166
Creating a Luminaire Source • 56
Creating a Measure from a XMP Result • 160
Creating a Polarization Plate Component • 33
Creating a Radiance Sensor • 82
Creating a Ray File Source • 49, 173
Creating a Rear Position Lamp with Light Guides • 220
Creating a Set of Features • 11, 58, 101, 123
Creating a Source Group • 56, 103, 104
Creating a Surface Source • 35
Creating a Thermic Surface Source • 51
Creating an Ambient Material • 20
Lesson 1
- Creating a Map • 182
- Creating the Macro • 192
- Opening Project • 166, 220
- Opening the Project • 188, 201, 212
- Preparing Data • 210

Lesson 2
- Applying Materials to a Product • 166
- Creating a 3D Texture • 183
- Creating a Polarizer Surface • 210
- Creating Optical Properties • 212
- Creating Parameters • 188, 201
- Declaring and Initializing Variables • 192
- Designing Light Guide Geometries with Optical Shape Design (OSD) • 221

Lesson 2.1
- Creating Light Guides • 221

Lesson 2.2
- Applying Material • 224

Lesson 3
- Adding Optical Properties to Materials • 166
- Analyzing Anti-glare Filter with Ray Tracing • 211
- Applying Optical Properties • 214
- Creating a VB Script • 189
- Creating Measures • 202
- Running an Interactive Simulation • 186
- Validating Photometric Performances with Light Modeling (LM) • 225
- Writing a Report • 194

Lesson 3.1
- Adding Optical Properties • 225

Lesson 3.2
- Adding Face Optical Properties • 225

Lesson 3.3
- Creating Sensors • 225

Lesson 3.3.1
- Creating Intensity Sensor • 225

Lesson 3.3.2
- Creating 3D Irradiance Sensor • 226

Lesson 3.3.3
- Creating 3D Energy Density Sensor • 227

Lesson 3.4
- Creating Simulations • 228

Lesson 3.4.1
- Creating Direct Simulation with LM • 228

Lesson 3.4.2
- Creating Direct Simulation with 3DED • 229

Lesson 3.5
- Running Simulations • 229
Lesson 4.7.2
Analyzing Observer Result • 243

Lesson 5
Creating a VB Script • 203
Creating an Interactive Simulation • 168
Creating an Irradiance SWIR Sensor • 216
Saving Optimization Result • 197
Writing the Optimization Script • 192

Lesson 6
Analyzing Simulation Result • 197
Creating a Reaction • 206
Creating a Surface Source • 168
Creating an Interactive Simulation for FOV analysis • 217
Launching the Optimization • 198

Lesson 7
Analyzing Results • 199
Creating an Inverse Simulation • 219
Creating an Irradiance Sensor • 169
Creating Optimizations • 206

Lesson 8
Creating a Direct Simulation • 171
Launching the Optimization • 207
Running & Analyzing an Inverse Simulation • 219

Lesson 9
Analyzing Results • 172, 208
LPF Result • 160
Luminaire Source • 56, 103, 104

M

Managing Documents • 11
Managing the Presets • 147, 148
Managing the Results Objects • 164
Managing the Results Sub-Tree • 164
Managing the Results within the Specification Tree • 164
Managing the Sensors Objects • 101
Managing the Sensors Sub-Trees • 101
Managing the Sensors within the Specification Tree • 60, 73, 82, 92, 93, 101
Managing the Servers within the OPTIS Distributed Computing • 158
Managing the Simulation within the OPTIS Distributed Computing • 158
Managing the Simulations Objects • 104, 107, 110, 113, 123
Managing the Simulations Sub-Tree • 123
Managing the Simulations within the Specification Tree • 104, 107, 110, 123
Managing the Source Objects • 57
Managing the Sources within the Specification Tree • 35, 40, 41, 42, 43, 46, 50, 51, 55, 56, 57
Managing the Sub-Trees • 44, 57
Mapping • 24
Mapping File • 24
Mapping Filtering • 26, 27, 28, 29
Matching Color between XMP Results and Calibrated Display • 16
Maximum Impact Number • 141
Meshing and Tessellation • 139
Monitor Color Calibration • 12
Multi-Threading • 6

O

Operation • 22
Optical Properties • 18
OPTIS Distributed Computing Restrictions • 11, 107, 109, 112
Other LCD Limitations • 14
Outdoor Light Filtering • 110, 112

P

Parameterizing the SPEOS Core Application • 156
Parameters of Surface Contribution Analyzer • 77, 87
Parameters of 3D Texture • 21
Parameters of a 3D Energy Density Sensor • 93
Parameters of a Direct Simulation • 107
Parameters of a LDC Component • 31, 32
Parameters of a Luminaire Source • 56
Parameters of a Measure • 160
Parameters of a Polarization Plate Component • 33
Parameters of a Radiance Sensor • 82, 85
Parameters of a Ray File Source • 50
Parameters of a Surface Source • 35, 36
Parameters of a Thermic Source • 51
Parameters of an Ambient Material • 20
Parameters of an Ambient Source with CIE Standard General Sky Type • 42
Parameters of an Ambient Source with CIE Standard Overcast Sky Type • 41, 42
Parameters of an Ambient Source with Environment Type • 46, 47
Parameters of an Ambient Source with Natural Light Type • 43
Parameters of an Ambient Source with Uniform Type • 40
Parameters of an Intensity Sensor • 73, 165
Parameters of an Interactive Simulation • 104, 105
Parameters of an Interactive Source • 54, 55
Parameters of an Inverse Simulation • 109, 110
Parameters of an Irradiance Sensor • 59, 60
Parameters of Export Simulation Rays as Geometry • 106
Parameters of Face Optical Properties • 18, 19
Parameters of Interpolation Enhancement • 120
Parameters of Surface Contribution Analyzer • 68, 95, 96
Parameters of Virtual BSDF Bench • 113
Pattern • 21, 31
Pattern Direction • 26, 27, 28, 29, 30
Pattern Scale • 22, 27, 28, 29, 30
Planar Illuminance • 62, 63
Polarization Plate Component • 32
Preset • 147
Presets • 105, 108, 110, 113, 146
Properties • 35, 51, 56, 57, 58
R
Radial Illuminance • 62, 64
Radiance Sensor • 82, 103, 104
Ray File Source • 49, 60, 103, 228
Reading the HTML Report • 159
Rectangular Mapping • 25
Results • 158
Results Management • 164
Results Properties • 164, 165
Running a Light Modeling Optimization • 179
Running a Simulation Using Network Update • 156
Running a Simulation when Using OPTIS Distributed Computing • 156
Running External Simulations on the Same Computer • 155
Running External Simulations Using External Update • 155
S
Semi-Cylindrical Illuminance • 62, 66
Sensor • 117
Sensors • 59
Sensors Management • 101
Sensors Properties • 60, 73, 82, 101
Setting Polarizer Surface • 209
Setting the Ambient Source as Background • 48
Setting the Determinist Calculation Properties for an Inverse Simulation • 126
Setting the Direct Simulation Properties • 125
Setting the Grid Properties • 103
Setting the IES Visualization Properties for a Luminaire Source • 59
Setting the IES Visualization Properties for a Surface Source • 58
Setting the Intensity Sensor Parameters • 102
Setting the Interactive Simulation Properties • 124
Setting the Inverse Simulation Properties • 40, 126
Setting the Monte-Carlo Calculation Properties for an Inverse Simulation • 110, 134
Setting the Parameters • 6, 165
Setting the Radiance Sensor Parameters • 102
Setting the Results and Simulations Options • 5
Setting the Simulation Properties • 22, 23, 92, 105, 107, 110, 116, 138
Setting the Thermic Source Parameters • 59
Simulation • 113
Simulation Properties • 104, 107, 110, 123, 124
Simulations • 103
Simulations Compatibility • 103
Simulations Management • 123
Smart Engine • 141, 229, 238, 239
Source • 116
Sources • 35
Sources Management • 57
Support • 21
Surface Contribution Analyser • 84, 85
Surface Contribution Analyzer • 63, 66, 75, 94