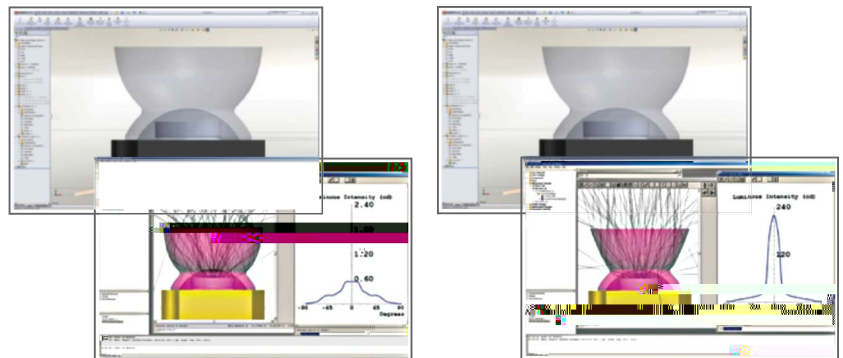


The LightTools® SOLIDWORKS® Link Module allows you to dynamically link SOLIDWORKS mechanical models to LightTools, where you can assign optical properties, optimize, and directly update your SOLIDWORKS designs.

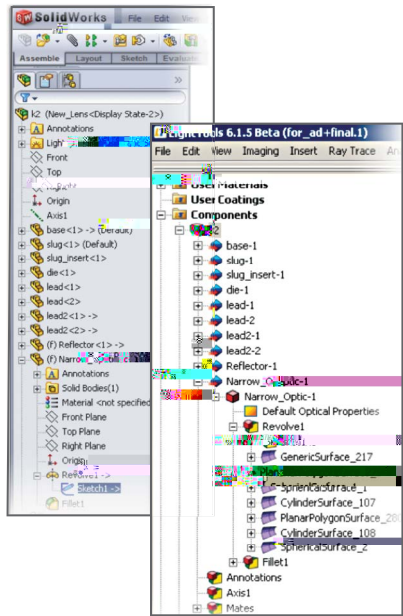
The SOLIDWORKS Link Module enhances the design and optimization capabilities of LightTools by allowing you to manipulate and modify SOLIDWORKS geometry in LightTools as if you were working with native LightTools geometry. Unlike a data exchange module, the SOLIDWORKS Link Module establishes a bidirectional connection between SOLIDWORKS and LightTools.

This unique feature enables you to update sketches and features in the SOLIDWORKS model from LightTools, see changes made in SOLIDWORKS to linked parts and assemblies from LightTools, and establish optical properties that are persistently defined in LightTools throughout the process. Because LightTools can update the model in SOLIDWORKS and receive the changes to the model from SOLIDWORKS without user interaction, you can also automatically refine SOLIDWORKS geometry using the LightTools Optimization Module.



Initial view

Final view



Powerful LightTools algorithms and improved data sharing with SOLIDWORKS create a dynamic duo of improved workflow and design accuracy.

With the SOLIDWORKS Link Module, you bring the CAD geometry into the model and set up its optical properties in LightTools once. Then, you can push one button to bring any subsequent updates from SOLIDWORKS into LightTools. LightTools automatically assigns the same optical properties to updated geometry that you defined for the original geometry, and the system is ready for analysis.

Alternatively, you can initiate changes to the SOLIDWORKS model from LightTools. The SOLIDWORKS feature-tree hierarchy is visible in the LightTools System Navigator, which allows you to update any driving numeric parameter (i.e., dimension or feature parameter) in the linked SOLIDWORKS model from the LightTools user interface. As with any other geometric entity in LightTools, you can also select SOLIDWORKS geometry to perform Boolean operations to create more complex geometry. This is especially useful if you need to use optical surface definitions, such as a polynomial asphere.

