

# Isight Component – ANSYS Workbench 12.1

The Isight ANSYS Workbench 12.1 component creates a direct link to ANSYS Workbench, enabling the modification of input parameters, execution of Workbench simulation, and extraction of output parameters.

ANSYS Workbench is a framework upon which the ANSYS suite of advanced engineering simulation technology is built, tying together the multi-physics simulation process. With bidirectional CAD connectivity, automated meshing, and parameter management, the ANSYS Workbench platform is an enabler for simulation driven product development.

## Component Specifications

- An Isight Activity component
- Provides support for all parameters defined in the ANSYS Workbench Parameter Set - this includes geometry, material properties, loads, deformations, stresses, strains, etc
- Supports various types of analyses such as Fluid Flow (Fluent), Static Structural, etc.
- Provides support to create in-model and output file parameters
- Allows user to supply custom python scripts to retrieve and update parameters which are not currently supported by the component and to perform other desired operations
- Parses input and output parameter information such as name, mode, value and type
- Allows user to filter the listed parameters by their name, mode and type
- Provides support for displaying/selecting available inputs and outputs by commands such as select, multi-select and select-all
- Compatible with Isight 4.5-1 and later releases
- Compatible with ANSYS Workbench 12.1 and 13.0 versions
- Available on Windows operating system

