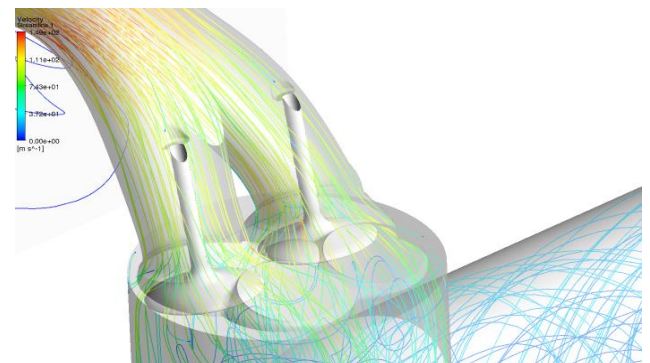


TECH BRIEF

SHAPE OPTIMIZATION IN ANSYS WITH CAESES®

CAESES® version 4.3 allows you to access and control the robust CAESES® geometry models within the ANSYS Workbench, to conduct simulation-driven shape optimization.

FRIENDSHIP SYSTEMS provides a CAESES®-ANSYS ACT app that allows you to plug-in any CAESES® geometry model into the user interface of the ANSYS Workbench.



Intake port optimization: How to find the best design candidate?

With just a few clicks you are ready to run large studies, such as design of experiments and formal optimizations including efficient surrogate-based optimizations – everything fully automated.

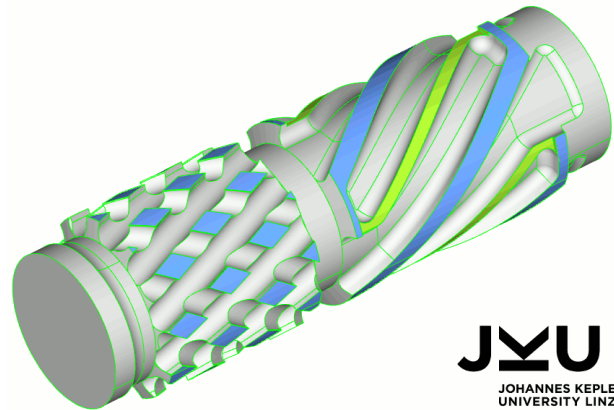
No matter which CAE task you have to solve (e.g. CFD and structural analysis) or which product you actually use (e.g. ANSYS FLUENT, ANSYS CFX or ANSYS Mechanical) – this new connection works for all of them.

OPTIMIZATION OF COMPLEX GEOMETRIES

There is no need to learn something new if you are use to the ANSYS Workbench. At the same time, you can directly benefit from the efficient and robust geometry models of CAESES® without any scripting efforts. Just plug them into the ANSYS Workbench and use drag & drop actions to make it run.

This new integration focuses on any kinds of geometries, from any application. CAESES® customers often deal with rather complex geometries that are exposed to flow such as ship hulls, propellers, rotors and stator blades, ducts, manifolds, exhaust systems, turbines and many more.

An example for a complex shape is a parametric shear head for polymer injection molding. Traditional CAD tools often fail to re-generate new design candidates of such complex parametric models in automated processes.

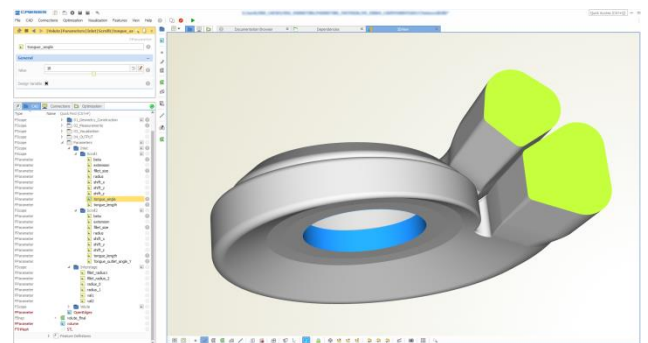


JKU
JOHANNES KEPLER
UNIVERSITY LINZ

Complex shear head geometry for polymer injection molding: Variable and 100% robust CAESES model for automated studies

PREPARATION IN CAESES

So how does this new connection actually work? First of all, you have to make sure that your CAESES® geometry model is ready with a set of design variables that control its shape. Export a so-called *.fsc control file of the setup via the CAESES® file menu (file > export > fsc file) – that's all you need.

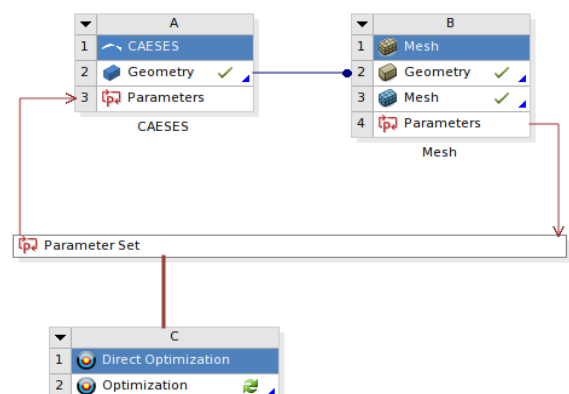


Geometry setup in CAESES

SHAPE OPTIMIZATION IN ANSYS

In the ANSYS Workbench, you have to install the CAESES®-ANSYS app (ACT extension) to make CAESES® available as a component in the ANSYS Workbench. Load the *.fsc file through the CAESES® component and update it.

The generated geometries get exported and are loaded into the ANSYS Workbench automatically using the export format you choose in CAESES®.



CAESES within the ANSYS Workbench

After the update of the CAESES® component in the ANSYS Workbench, the design variables of the geometry are automatically shown in the parameter set. New design candidates can now be generated by changing these parameters, either manually or by optimization tools (in-built strategies from ANSYS, optiSLang etc.).