

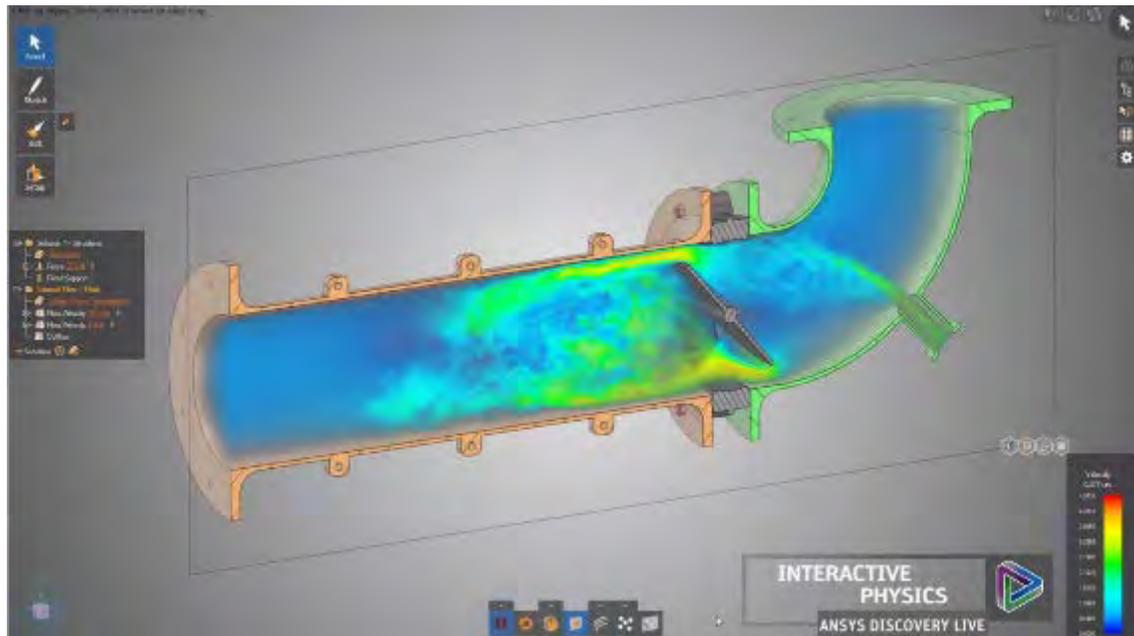


ANSYS Discovery Live- Getting Started

Every engineer deserves the power of Discovery

ANSYS Discovery Live provides instantaneous simulation, tightly coupled with direct geometry modeling, to enable interactive design exploration and rapid product innovation. It is an interactive experience in which you can manipulate geometry, materials and physics, and instantaneously see 3-D results.

Currently available for download as a technology preview, ANSYS Discovery Live fits into the 3-D design family of products and is complementary to ANSYS SpaceClaim and ANSYS AIM.



Any questions, or to hear about other technologies from ANSYS, please contact:
ANSYS Account Manager, dan.hartman@padtinc.com (480) 813-4884

Instructions for Accessing the Free Trial Beta License

The software is provided as an Evaluation License at no charge for the duration of the preview period, which ends February 1, 2018.

Accessing:

- Go to: <https://account.ansys.com/discoverylive?source=PADT>
 - Follow instructions for setting up your Discovery Live account
 - Choose to download and run Discovery Live Local (must meet hardware requirements below) or the Cloud Trial Option
-

ANSYS Discovery Live Hardware Requirements:

ANSYS Discovery Live relies on the latest GPU technology to provide its computation and visual experience. To run the software, you will require:

- A dedicated NVIDIA GPU card based on the Kepler, Maxwell or Pascal architecture. Most dedicated NVIDIA GPU cards produced in 2013 or later will be based on one of these architectures.
- At least 4GB of video RAM (8GB preferred) on the GPU.

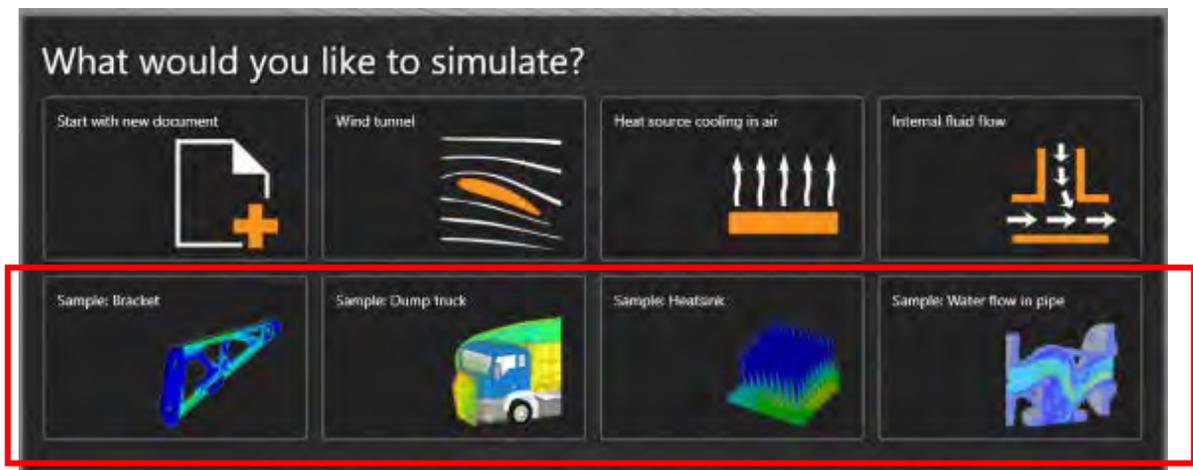
Also, please ensure you have the latest driver for your graphics card, available from [NVIDIA Driver Downloads](#). You can also refer to the post on [Graphics Performance Benchmarks](#).

Performance of Discovery Live is less dependent on machine CPU and RAM. A recent generation 64-bit CPU and at least 4GB of RAM will be sufficient.

If you do not have a graphics card that meets these specifications, the software will not run. However, you can try ANSYS Discovery Live through an [online cloud-based trial](#), which requires only an internet browser and a reasonably fast internet connection.

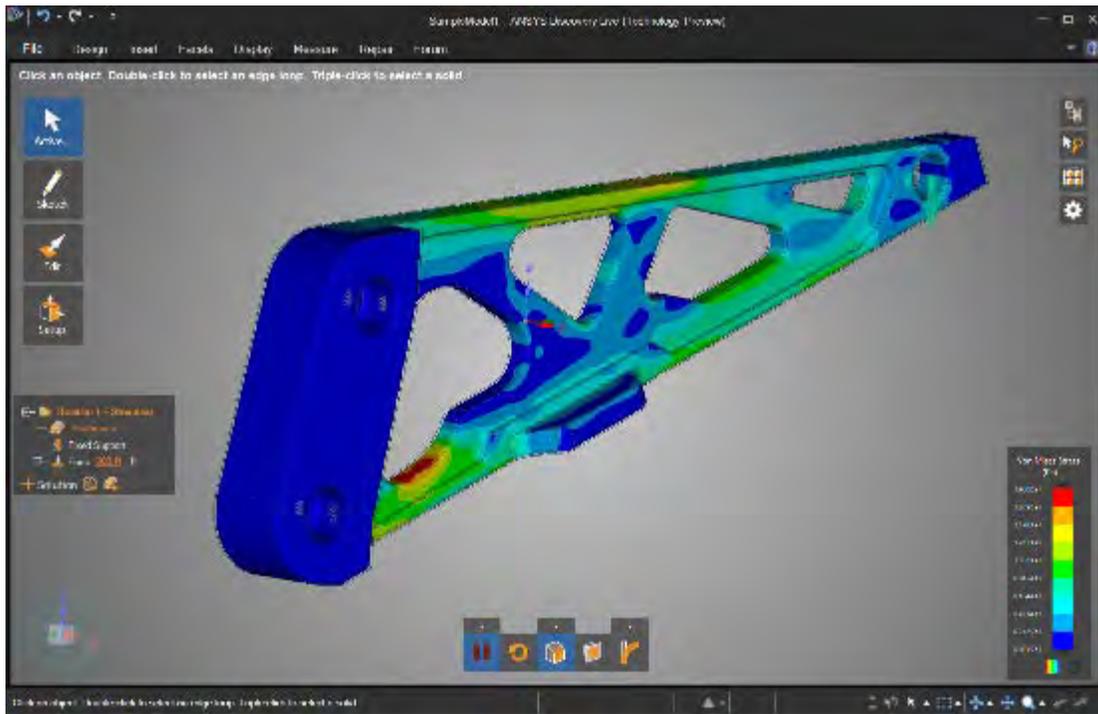
ANSYS Discovery Live Quick Start Instructions

1. After opening Discovery Live, you will be greeted with the starting panel. Recommendation is to start with one of the pre-built examples from bottom row



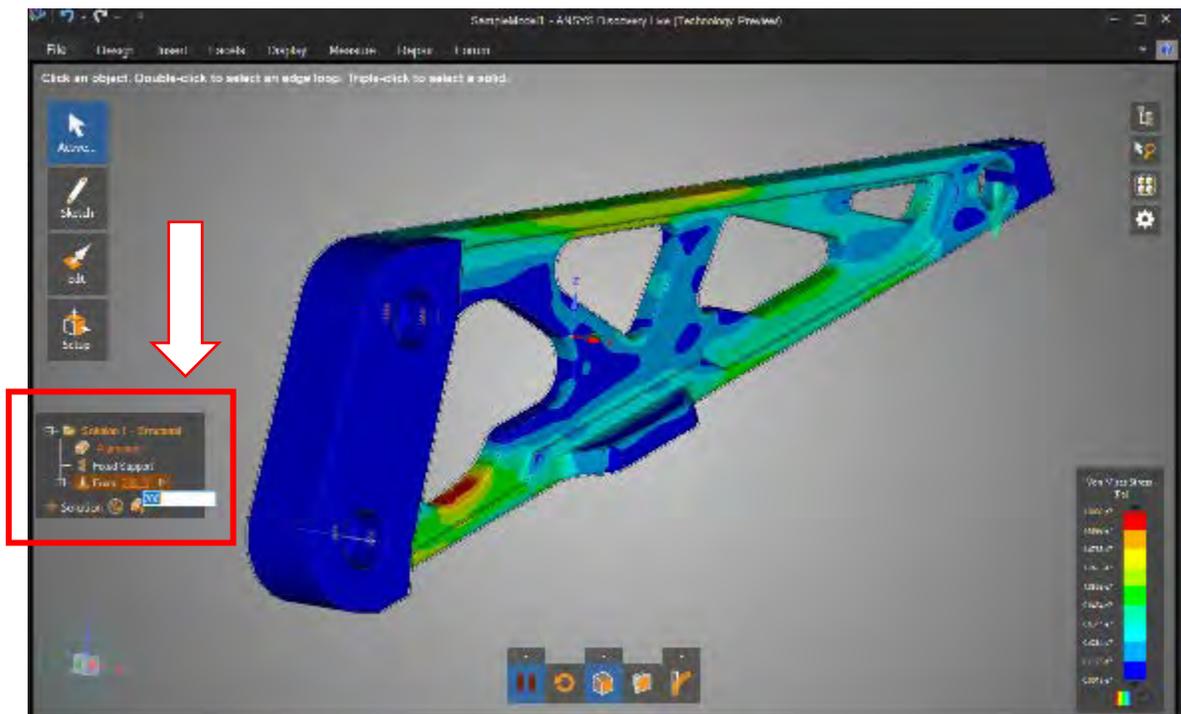
Starting Panel in ANSYS Discovery Live

2. For example, opening the “Sample: Bracket” file will open the geometry and instantaneously solve the model



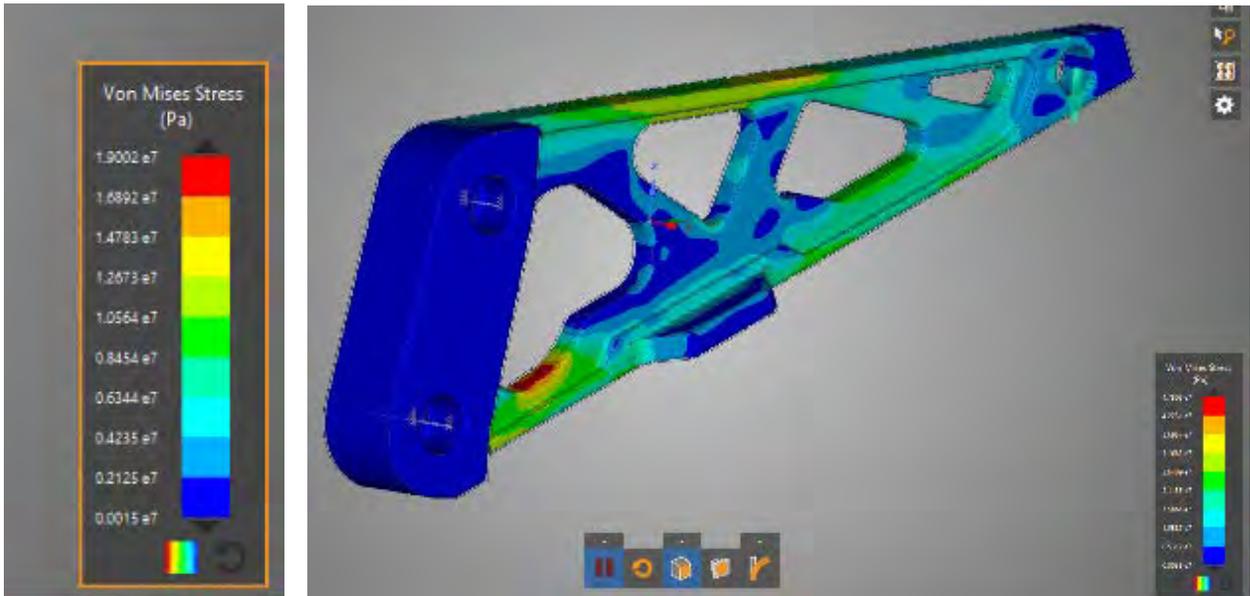
1st look after opening Sample:Bracket file

- Using this example, you can change the force or any of the structural boundary conditions on the left side of the window



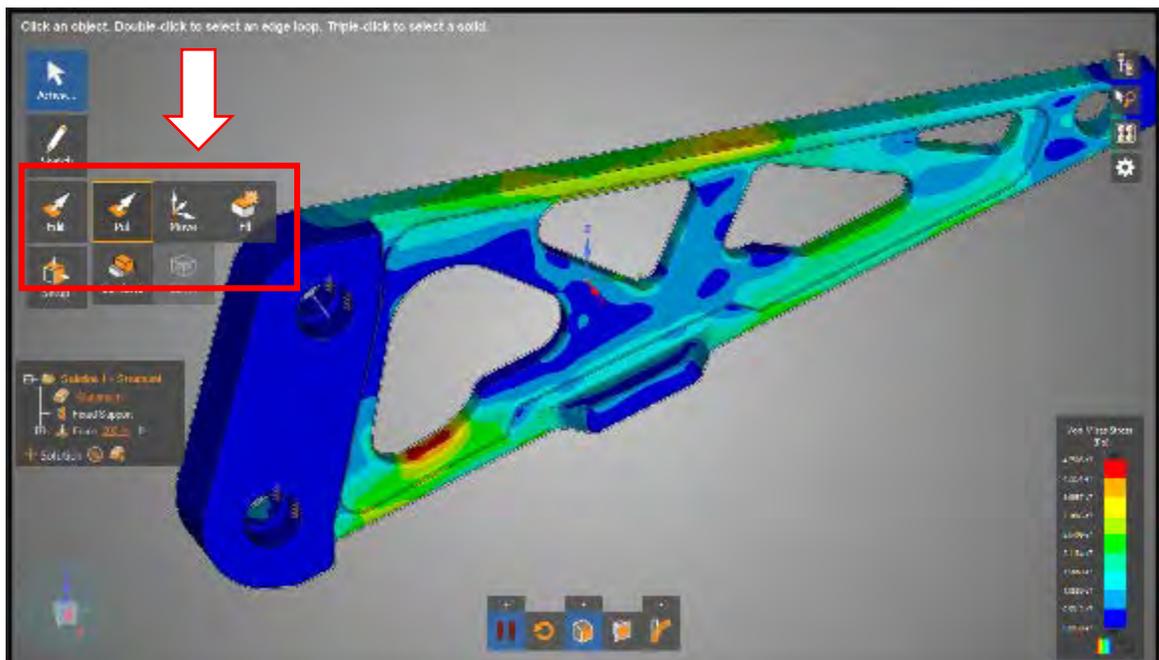
Boundary conditions can be modified on left side of window

- As soon as you change the value, results will immediately shift to reflect the new load



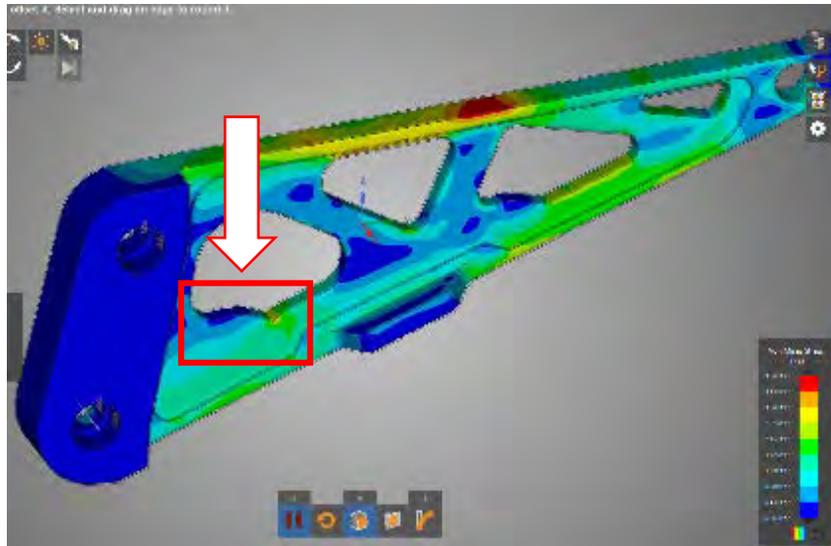
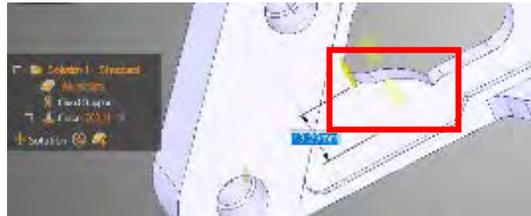
Left: Initial Results | Right: Modified Force Results

5. To modify geometry, click on “Edit” on left side and choose from one of the modification tools such as **Pull**



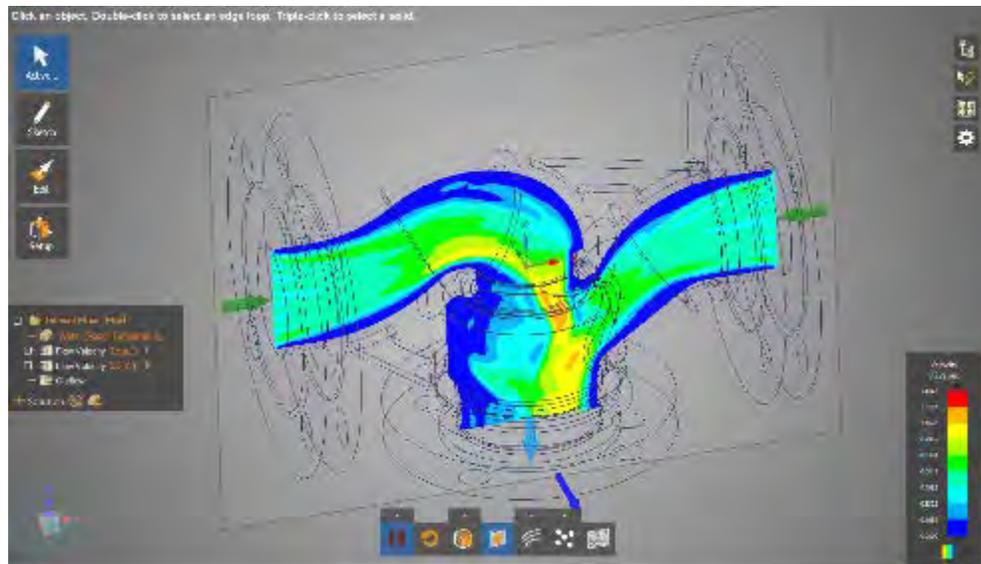
Edit → lets you change geometry and have results change accordingly

6. By Pulling this face, we can see the sharp stress disappear immediately



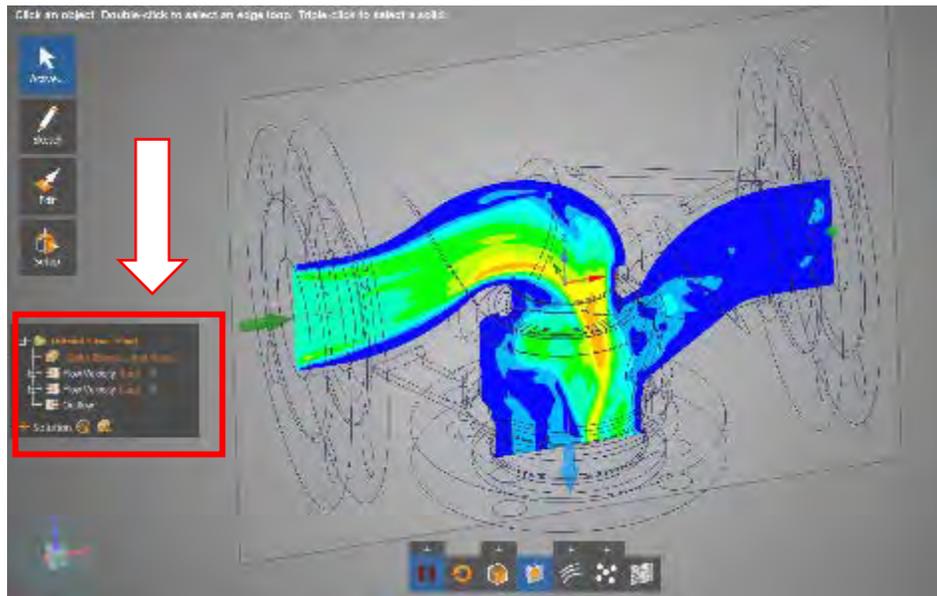
Pull lets you extrude a selected face and as soon as movement is made, results change to reflect new geometry

7. In the same way you can use some of the other sample geometries to interact with the results live



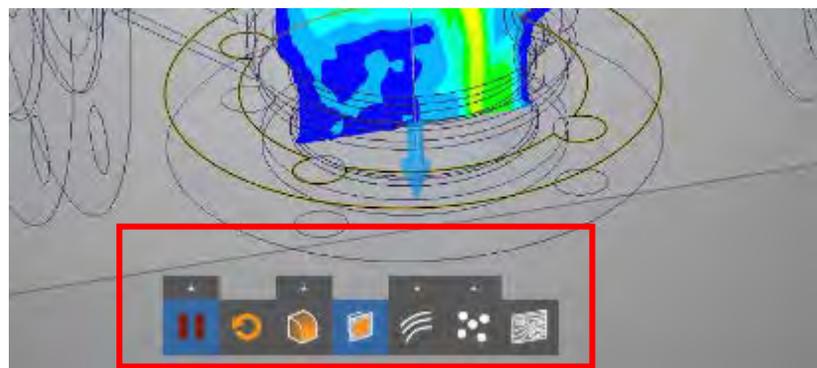
Sample: Water Flow in Pipe

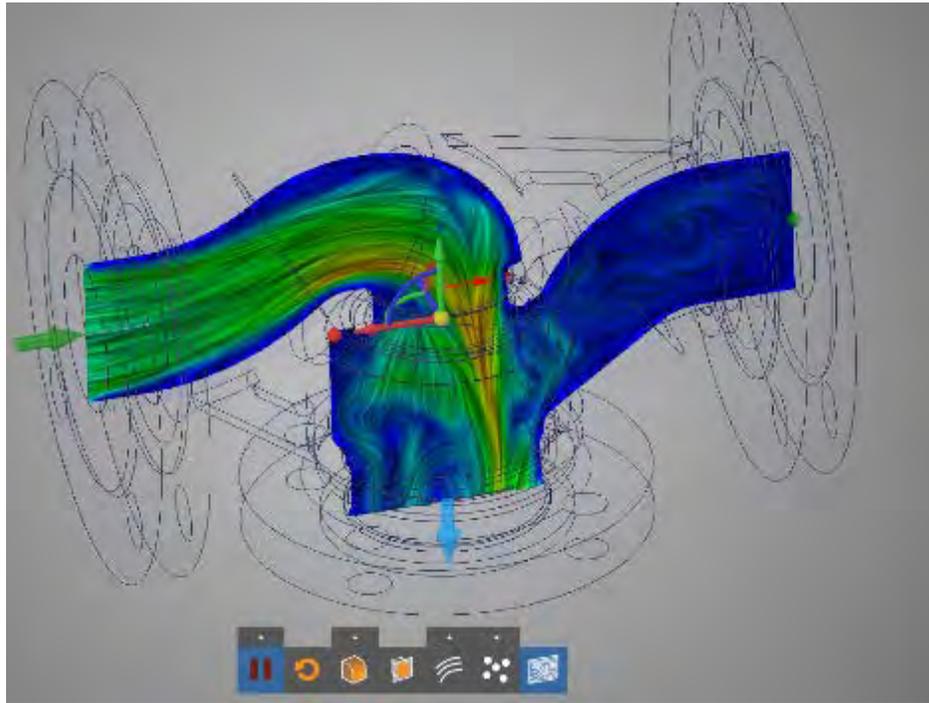
8. For fluid analysis you can change the boundary conditions the same way, by modifying the values on the left



Changing boundary conditions instant changes the velocity field

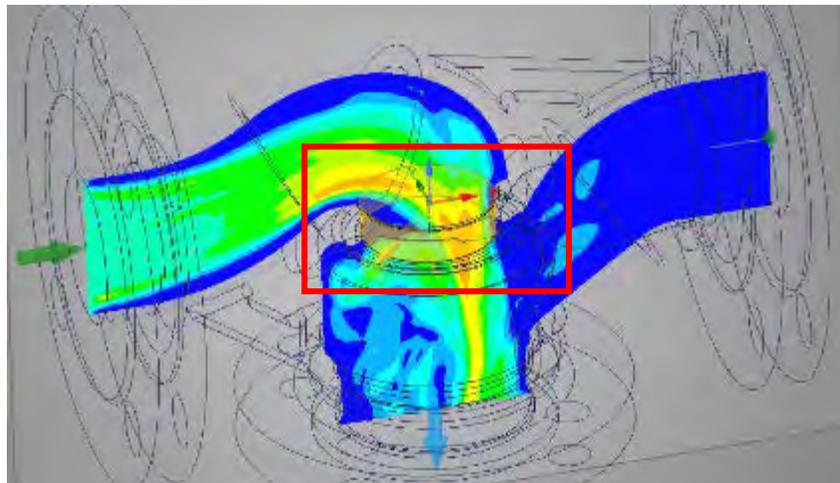
9. You can see these options at bottom of screen to change to different result views. For example looking at LIC instead

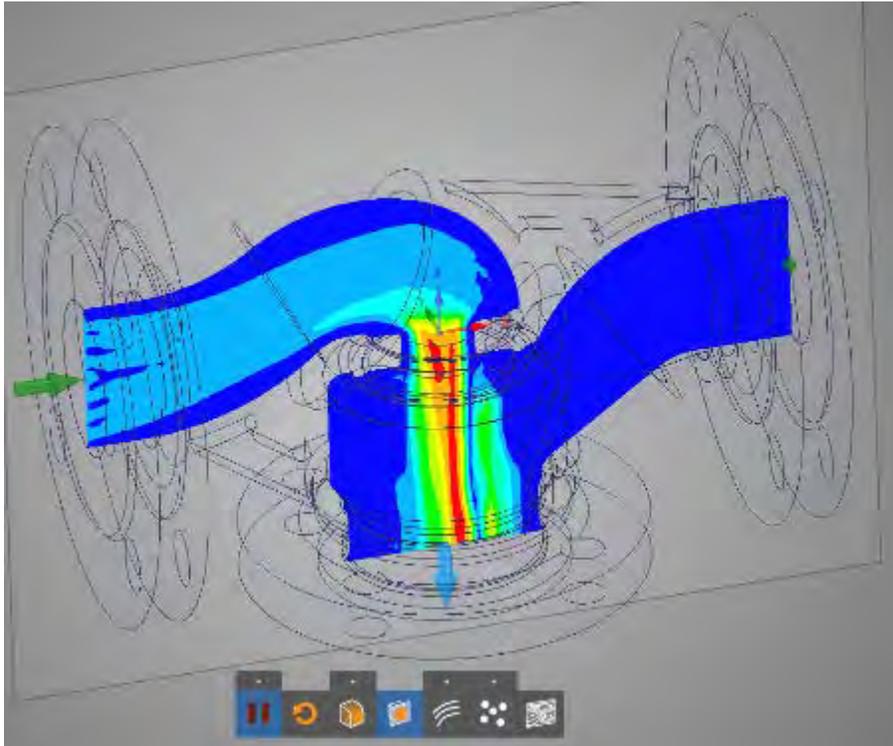




TOP: Use bottom buttons to change result views | BOTTOM: Shows LIC map of fluid results

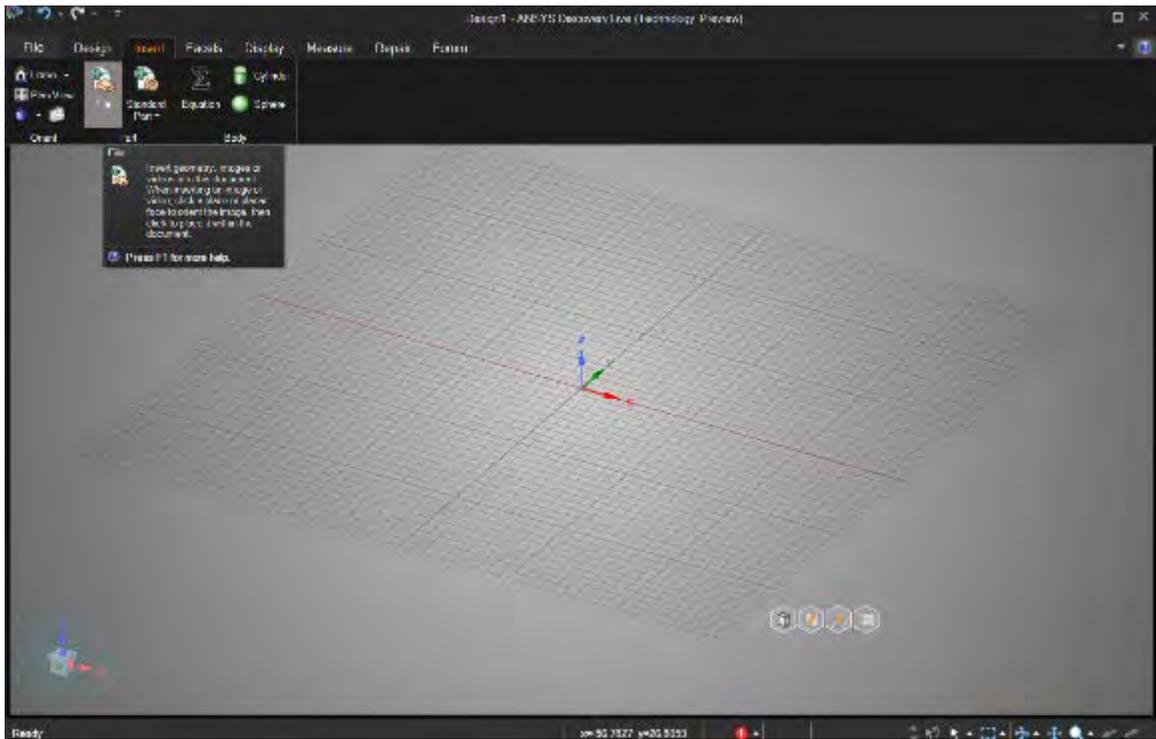
10. In the same way that geometry was changed using the Edit options on the left panel, users can do the same for fluid analyses with instant results





TOP: Using geometry modification tools to change neck diameter | BOTTOM: Shows results as a function of the geometry change

11. Users can certainly also create new geometry from scratch or import geometry from other sources



By selecting “Start With New Document” in Start panel, users can create new geometry or import existing geometry

