AUTODESK® INVENTOR™

Benefits of Using Autodesk Inventor Simulation Suite in Engineering Design

Over the past couple of decades, there have been several dedicated finite element analysis (FEA) and motion analysis software applications available to engineers. These comprehensive technologies have been used primarily by analysis and simulation engineering experts. Major drawbacks of such dedicated software include the advanced knowledge required and their lack of integration with 3D computer-aided design (CAD) software. Consequently, designers continue to work in isolation from one another, resulting in duplication of work, increased design change time and cost, and longer time to market.
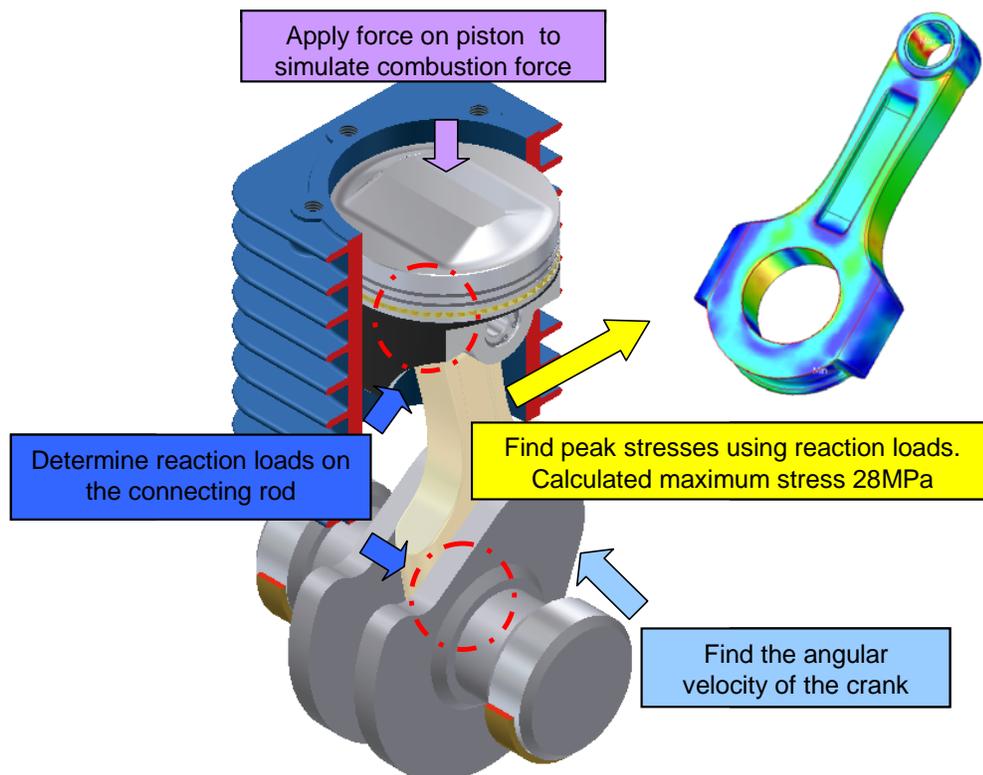
The Autodesk® Inventor™ Simulation Suite software application addresses these drawbacks. Built on the Autodesk Inventor application, it links seamlessly with Autodesk Inventor 3D model and requires significantly less training to drive its powerful analysis and simulation technologies. In brief, this new technology enables engineers to build and analyze digital prototypes that enable designers to develop competitive designs more efficiently and cost effectively.

Autodesk Inventor Simulation Suite is a must for designers and engineers looking for a tool to help them resolve design problems, reduce failures and warranty costs, and accelerate turnaround time. It is the perfect solution for engineers and designers involved in the design and manufacture of:
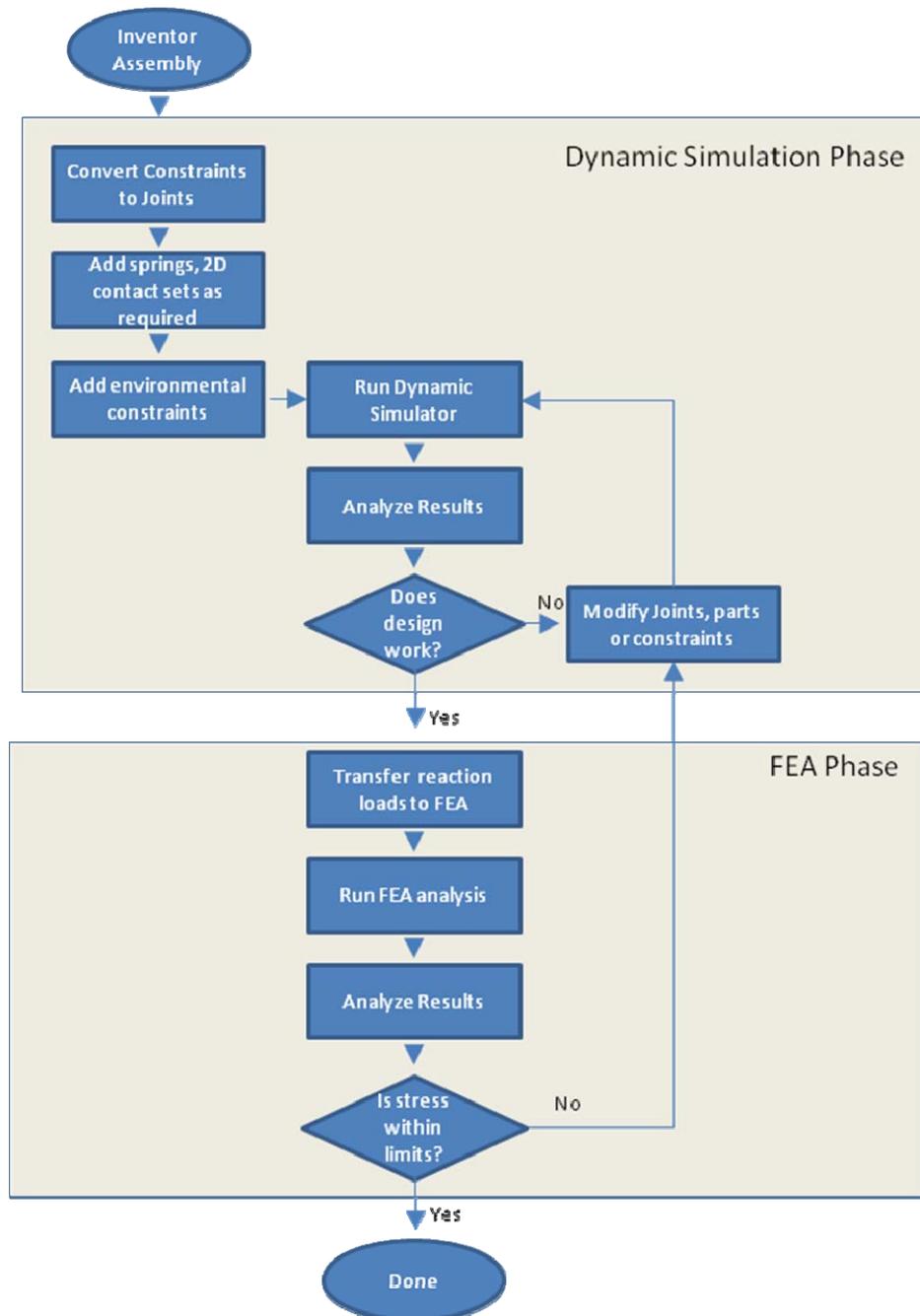
- Industrial machinery
- Medical devices
- Automotive
- Aerospace
- Equipment for construction
- Consumer products

Introduction

During a typical design process, designers must answer a series of questions such as: Do the parts fit together? Do the parts move well together? Is there interference? Do the parts follow the right path? Most of these questions can be addressed by 3D CAD and rendering software, but other questions cannot. For example, designers may need to know the machinery time cycle. Is the actuator powerful enough? Is the link robust enough? Can we reduce weight? These types of questions have traditionally been answered by building a working physical prototype or even a series of prototypes—a time-consuming and costly proposition. A more cost-effective alternative is to leverage the digital prototype by using Autodesk Inventor Simulation Suite software. This software enables designers to automatically convert assembly constraints to mechanical joints; to apply external forces, including gravity; and to measure the effects of contact friction, damping, and inertia. From this information the software calculates reaction forces, velocities, acceleration, and much more. Reaction forces can then be used to drive FEA analysis, thereby reducing guesswork and risk. Autodesk Inventor Simulation Suite software is helping designers build optimized and competitive products.

Apply force on piston to simulate combustion force

Determine reaction loads on the connecting rod

Find peak stresses using reaction loads. Calculated maximum stress 28MPa

Find the angular velocity of the crank

Autodesk® Inventor™ Simulation Suite software combines a simulator for motion dynamics with a powerful linear finite element stress analysis application. To understand the performance of a proposed design, a designer starts by running the dynamic simulation tool to calculate the dynamic performance. The results of the simulation represent the positions, velocities and accelerations of the mechanism throughout its operation cycle. It also includes the reaction forces at each joint. These forces are then used as the input parameters for the second phase in which finite element stress analysis is used to compute the internal stresses in those highly loaded parts. This process is summarized in the following diagram:

```
                    ┌─────────────┐
                    │  Inventor   │
                    │  Assembly   │
                    └─────────────┘
                           │
    ┌──────────────────────────────────────────────────────────┐
    │                            Dynamic Simulation Phase        │
    │  ┌──────────────────┐                                      │
    │  │ Convert Constraints│                                    │
    │  │    to Joints      │                                     │
    │  └──────────────────┘                                      │
    │           │                                                │
    │  ┌──────────────────┐                                      │
    │  │ Add springs, 2D  │                                      │
    │  │ contact sets as  │                                      │
    │  │    required      │                                      │
    │  └──────────────────┘                                      │
    │           │                                                │
    │  ┌──────────────────┐    ┌──────────────┐                 │
    │  │Add environmental │───▶│ Run Dynamic  │◀──────────┐     │
    │  │   constraints    │    │  Simulator   │           │     │
    │  └──────────────────┘    └──────────────┘           │     │
    │                                  │                   │     │
    │                          ┌──────────────┐            │     │
    │                          │Analyze Results│           │     │
    │                          └──────────────┘            │     │
    │                                  │                   │     │
    │                              ◇ Does ◇   No  ┌──────────────────┐ │
    │                              ◇design ◇─────▶│Modify Joints, parts│ │
    │                              ◇ work? ◇      │ or constraints    │ │
    │                                  │          └──────────────────┘ │
    └──────────────────────────────────────────────────────────┘
                                  │ Yes
    ┌──────────────────────────────────────────────────────────┐
    │                                        FEA Phase           │
    │                          ┌──────────────┐                 │
    │                          │Transfer reaction│              │
    │                          │ loads to FEA  │                │
    │                          └──────────────┘                 │
    │                                  │                         │
    │                          ┌──────────────┐                 │
    │                          │Run FEA analysis│               │
    │                          └──────────────┘                 │
    │                                  │                         │
    │                          ┌──────────────┐                 │
    │                          │Analyze Results│                │
    │                          └──────────────┘                 │
    │                                  │                         │
    │                              ◇Is stress◇   No             │
    │                              ◇ within ◇──────────▶        │
    │                              ◇ limits?◇                   │
    └──────────────────────────────────────────────────────────┘
                                  │ Yes
                            ┌─────────────┐
                            │    Done     │
                            └─────────────┘
```

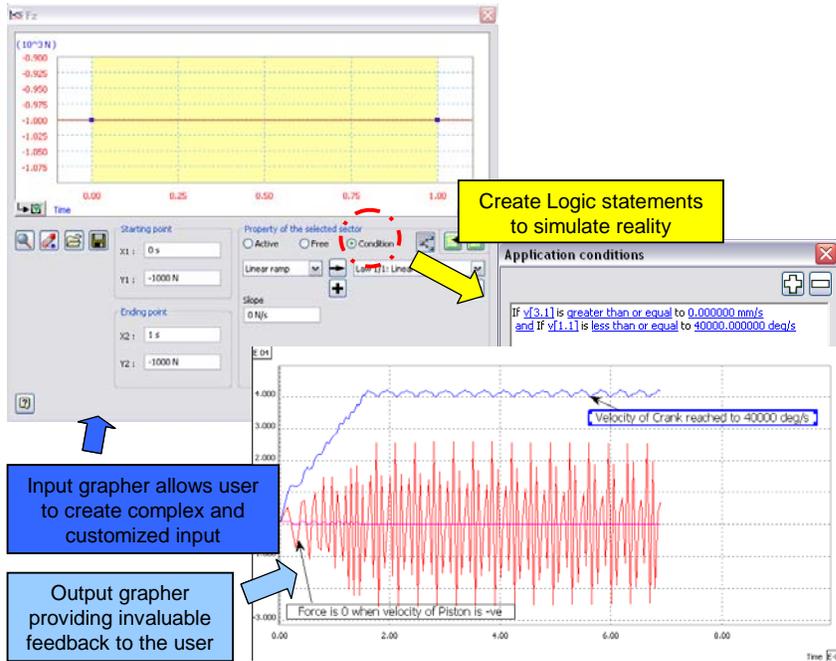# Design Simulation and Analysis of a Connecting Rod

After a designer has designed and assembled the initial 3D model in Autodesk®
Inventor™ software, it takes just one click to transfer the 3D model into Autodesk Inventor
Simulation Suite. Additionally, the designer can easily work seamlessly between the
design and simulation environments.

In the simulation environment, the first step is to convert the assembly constraints into
mechanical joints. This can be done in several ways, but the most efficient method is to
use the Automatic Update of Translated Joints feature as illustrated in the following
graphic. This tool converts all assembly constraints in a few seconds.
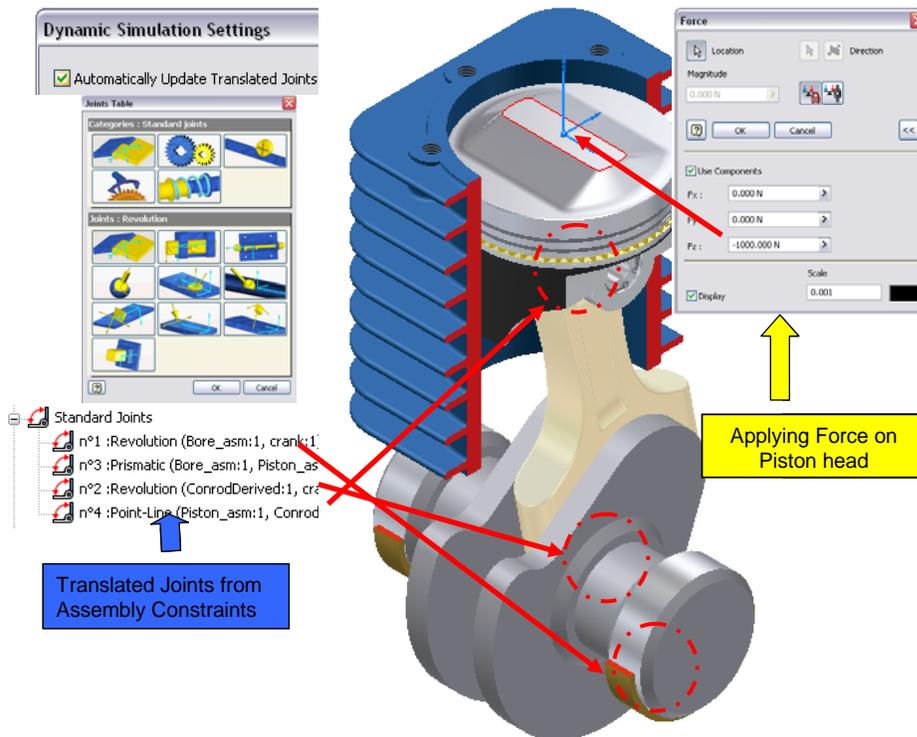
The next step in the simulation is to define any additional specialized motions,
environmental constraints, and the driving forces. In the piston assembly illustrated above,
a driving force is exerted on the piston to mimic the combustion force that occurs when the
driver applies pressure on the accelerator. The more acceleration, the more combustion
force, hence the faster the vehicle moves. Using the powerful Input Grapher feature, the
designer can simulate the input force with ease. For example, the designer creates a logic
statement or condition that switches off the force on the upward stroke of the piston when
the crank (engine) achieves a certain speed, as shown in the following graphic.

Create Logic statements to simulate reality

Input grapher allows user to create complex and customized input

Output grapher providing invaluable feedback to the user

After entering the correct environmental constraints and joints, the designer runs the simulation and analyzes the results using the Output Grapher, as shown in the following graphic. In addition to displaying standard parameters, the designer can also create custom parameters in order to effectively analyze the simulation.

One of the most useful parameters that designers can analyze and display is the reaction forces at the joints. Traditionally, this type of analysis has required time consuming calculations or measurements from costly physical prototypes.
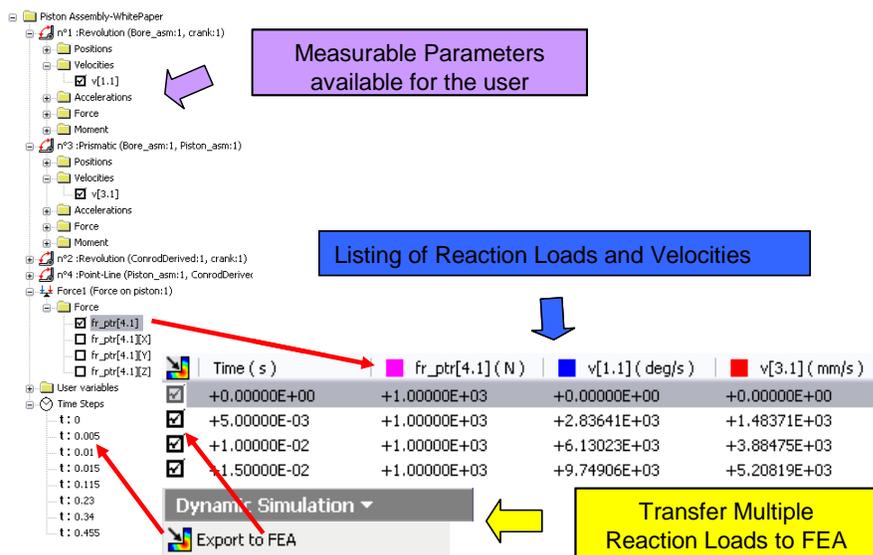
Measurable Parameters available for the user

Listing of Reaction Loads and Velocities

Transfer Multiple Reaction Loads to FEA

Time (s)	fr_ptr[4.1] (N)	v[1.1] (deg/s)	v[3.1] (mm/s)
+0.00000E+00	+1.00000E+03	+0.00000E+00	+0.00000E+00
+5.00000E-03	+1.00000E+03	+2.83641E+03	+1.48371E+03
+1.00000E-02	+1.00000E+03	+6.13023E+03	+3.88475E+03
+1.50000E-02	+1.00000E+03	+9.74906E+03	+5.20819E+03

Using Dynamic Simulation, the designer can also transfer multiple time step loads to the stress analysis environment, as illustrated in the preceding graphic. This step provides significant benefits by reducing guesswork and risk. These loads are automatically transferred and applied to the correct bearing load faces of the component being analyzed. Because the loads are transferred automatically, designers do not need to apply loads and constraints as they would when using FEA analysis in isolation. With the loads successfully applied, the designer clicks the Run button to start the analysis. There is no need to specify any mesh sizes, as this is done automatically. The analysis provides a graphic display that shows the areas of peak stress using a color coded scale. This makes it very easy to identify the areas of the part that will experience high stress. By analyzing these results, the designer can make design changes to redistribute the stress and reduce peak stresses. It may also be possible to reduce weight without increasing the peak stresses in the part. Using these techniques, designers can quickly develop parts that minimize material usage while at the same time improving the efficiency of the final product.
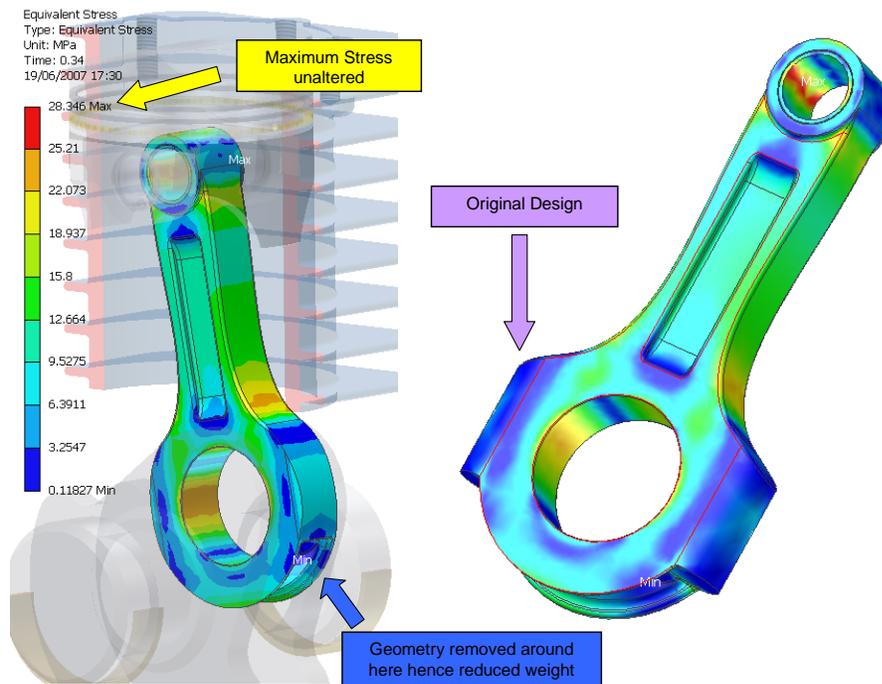
Conclusion

Autodesk Inventor Simulation Suite addresses the shortcomings of traditional, dedicated, FEA and motion analysis solutions by providing easy to use tools that work directly with the Inventor 3D model. The resulting ease of use gives designers a true digital prototype that can be used to experiment with different design options. At the same time, it can increase confidence that the performance of the final design will meet the customer's expectations. The ability to predict and transfer reaction forces to the stress analysis environment allows designers to leverage the digital prototype to optimize part geometry while reducing the need to build costly physical prototypes.

With Autodesk Inventor Simulation Suite, designers can

- Get to market faster
- Decrease warranty costs
- Improve product quality

Autodesk®