

# Assess Design Performance with Finite Element Analysis

To compete in today's global manufacturing supplier market, companies are challenged to:

- Shorten development cycles and cut manufacturing costs
- Increase innovation and creativity
- Produce better-quality goods

Incorporating FEA (finite element analysis) into the product design process can help speed production. By trimming development cycles, companies can reduce a product's time to market and quickly adapt to new market requirements. In particular, FEA helps minimize the prototyping phase of development.

For example, if the table of an industrial machine is in resonance because its fundamental frequency is close to that of the drive, the manufacturer has to modify the underlying properties of that table. Typically, that means time and money spent correcting the design flaws on a prototype.

But FEA testing completed early in the design phase can reveal potential trouble spots before the prototype is built. Prospective problems can be corrected in the design to minimize the changes required in prototype -- accelerating both product development and production. Autodesk Inventor Professional incorporates FEA testing to accomplish viable designs more quickly and cost-effectively.

## Get Acquainted with FEA

FEA requires you to establish boundary conditions - the part materials and a definition of how the part operates - and then the software simulates the behavior of a component in its physical environment, taking into account how forces will affect the part and in what way the part is supported.

FEA is completely integrated into the Autodesk Inventor Professional interface, so you can perform testing within the design application. If you aren't familiar with Autodesk Inventor Professional FEA procedures, use the predefined order in the panel bar to simplify the process. Test a few components whose results you already know, and try to verify these known results using the FEA tools.

The FEA module displays the design to be tested -- in this example, a simple bracket -- along with related data and the functions you can perform (figure 1). The browser or tree structure summarizes all input data and facilitates calculation management. The panel bar offers direct access to basic features. Its top-to-bottom organization helps guide less-experienced users and ensures that all required information is entered.

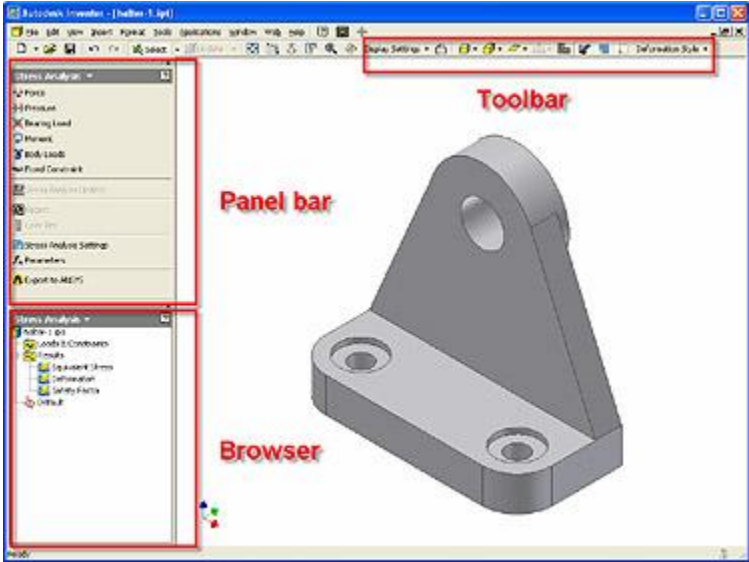


Figure 1. FEA module display elements show input data and features at your disposal.

### Prepare a Part Design for Analysis

To get started, click on Stress Analysis in the Autodesk Inventor Professional in the panel bar (figure 2).

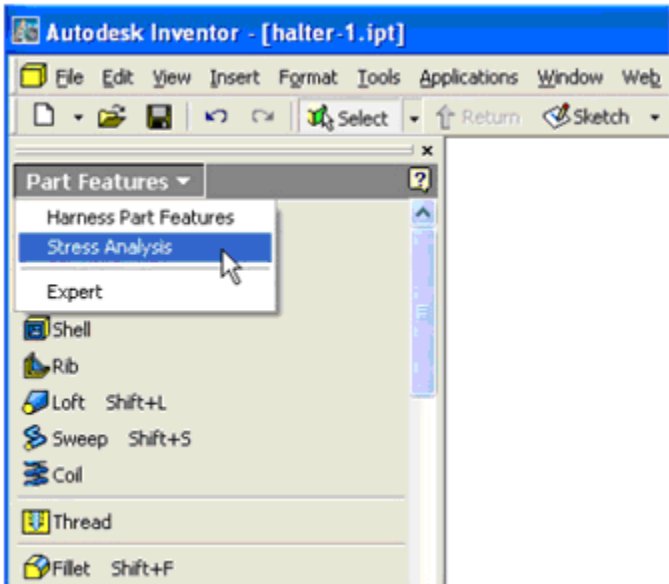


Figure 2. Choose Stress Analysis to start the FEA module.

The first step in stress analysis is to define the part's material. Specify one of the predefined materials and assign it to the component (figure 3)

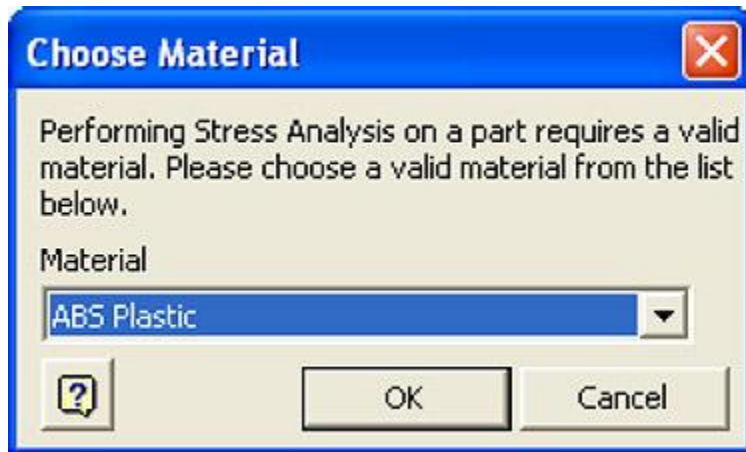


Figure 3. Define the part's composition by selecting a material.

Next, define loading by selecting the appropriate features in the panel bar to define forces, pressures, moments and so forth. For example, let's apply a downward force of 20,000 newtons to the center hole in the bracket. Select Force in the panel bar and choose the geometry on which the force will act: This can be one or more surfaces, edges or points. The small arrow symbol in the center hole shows the default direction of the force. You can reverse the direction of force by using the Direction icon in the dialog box (figure 4).

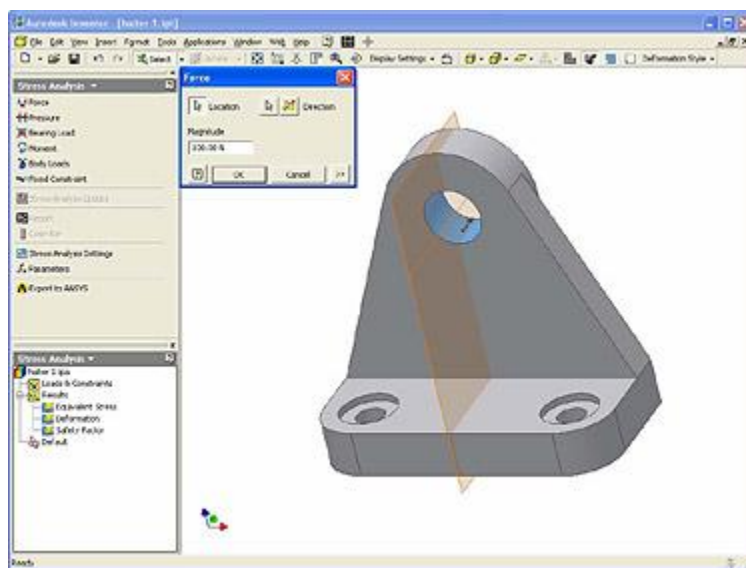


Figure 4. Define the force to be applied to the part, and its direction.

Now a support must be defined that will maintain the bracket in position. Select Fixed Constraint in the panel bar and highlight the lower supporting surface of the bracket to create a fully fixed support defined for that specific area (figure 5).

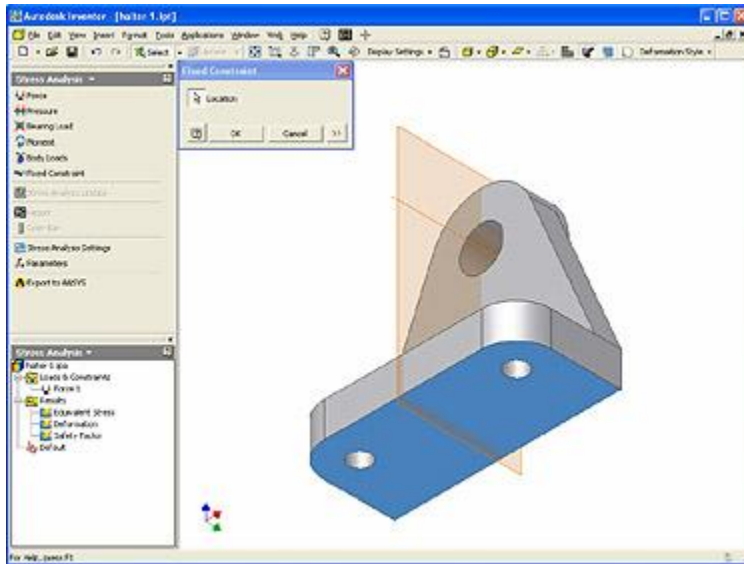


Figure 5. Specify a support that will hold the bracket in position.

## Analysis Automatically Shows Areas for Improvement

Choose Stress Analysis Update in the panel bar, and an indicator will appear to show the progress of the test. Once the operation is complete, results are mapped onto the part model so you can see performance at a glance (figure 6). You can select results for equivalent stress, deformation and safety by double-clicking the result icon next to the result name, in the FEA module browser.

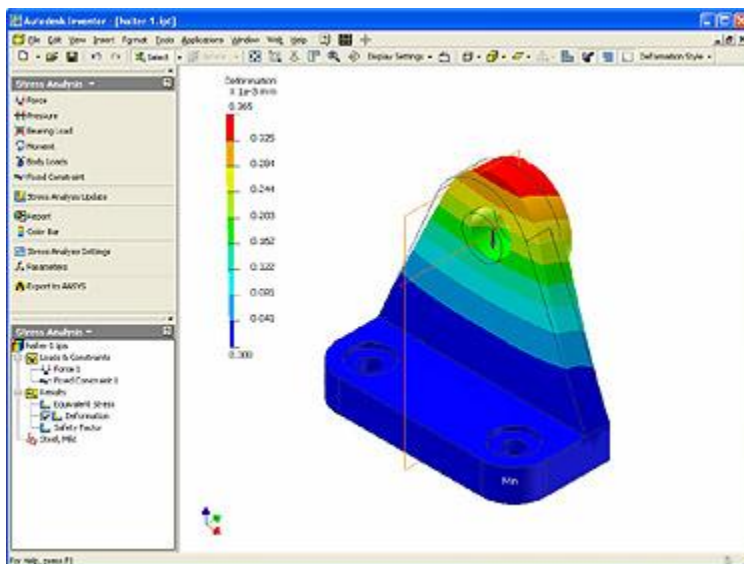


Figure 6. FEA performance results are displayed graphically.

## **Final Thoughts**

When conventional wisdom holds that prototyping accounts for as much as 25% of the entire development cycle, virtual testing offers an efficient and cost-effective alternative. The FEA module integrated into Autodesk Inventor Professional helps you validate basic design properties in the early phases of the drafting and development process. As a result, you can drastically reduce the number of required physical prototypes, and produce these physical models with greater confidence in their likely performance.