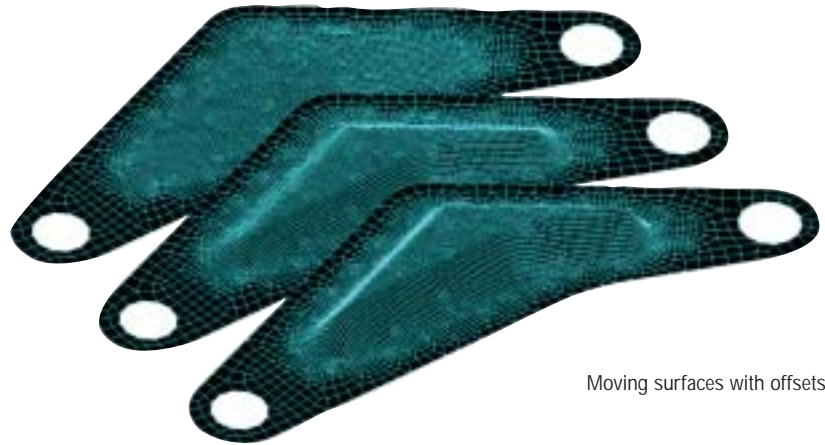


Product Benefits

- ▶ Quickly makes a static FEA model parametric and modifiable for optimization or "what if" studies
- ▶ Parameterization of a static FEA model is easy, fast, and very flexible
- ▶ All changes are done at the nodal coordinate level, saving tremendous amounts of time over traditional methods
- ▶ Easily imports and modifies large models, including million degree of freedom aerospace airframes and automotive bodies in white
- ▶ Doesn't require a CAD license or a CAD expert's time to make modifications to a model
- ▶ Provides an almost unlimited amount of flexibility to transform and move surfaces, edges, vertices and dramatically change the shape of an FEA model
- ▶ Saves a tremendous amount of time in the design/simulate process by using an existing mesh rather than requiring the user to generate a new mesh for each design exploration cycle
- ▶ Can be used with all analysis types, structural analysis, modal analysis, Computational Fluid Dynamics, electromagnetics, acoustics, even Multiphysics
- ▶ Includes an automatic, global element shape smoothing algorithm that controls element shape quality throughout the morphing process
- ▶ Can be used with models written for all major structural analysis formats, i.e. ANSYS, NASTRAN, Patran, ABAQUS
- ▶ Can be used with models written for many major CFD codes, i.e. Fluent, Star-CD

Time is Money! What makes money – particularly to hyper-competitive companies that use engineering as a key competitive weapon – is to find ways to reduce wasted time, to streamline a process and to develop the best design faster than anyone else can. “Time to Best Design” now is a closely contested race.

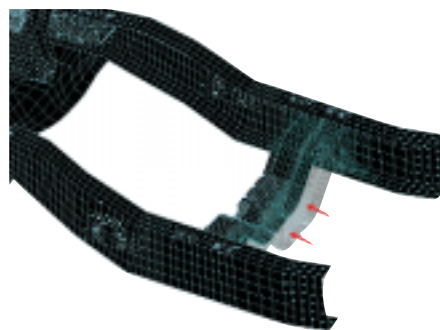
ANSYS ParaMesh is all about process streamlining. Inserting ANSYS ParaMesh into a typical product development process can sometimes halve the amount of time it takes to get to the best, simulation-proven design. Those kind of claims are jaw-dropping and difficult to believe, until you see ANSYS ParaMesh in action. ANSYS ParaMesh allows you to work directly with your existing analysis model. By working only with the nodal coordinates and a variety of transformations, ParaMesh is able to morph your existing mesh into a wide array of new designs.



Moving surfaces with offsets

Why ANSYS ParaMesh?

- ▶ Very effective with legacy models; many projects re-use existing models
- ▶ Very efficient with large models that are difficult to modify by any method
- ▶ To drastically reduce the amount of time it takes you to perform a design iteration
- ▶ To give you an advantage over those that use only the traditional process



Translating cross-member with offsets

Applications

- ▶ Perform rapid modifications of a design (evolution of existing designs)
- ▶ Perform concept analysis at the CAE level
- ▶ Explore many design alternatives
- ▶ Perform shape optimization without CAD models
- ▶ Perform easier and more accurate Design of Experiments (DOE)

ParaBatch

ParaBatch is the “batch only” version of ParaMesh, which means that the GUI (Graphical User Interface) is not active. The mode of operation allows you to

Product Features

Transformations

- ▶ Group definitions
 - Translations, rotations and scaling on vertices, edges or surfaces
 - Projection on target surfaces (with optional Geometry Module)
 - Bounding box deformations

Parameterization

- ▶ Each transformation is associated to a parameter
- ▶ Mesh generation for any parameter combination
- ▶ Analysis of the mesh quality before and after modifications

ParaMesh Import/Export

- ▶ FE Meshes
 - ABAQUS
 - ANSYS
 - NASTRAN
 - PATRAN
 - Fluent
 - Star-CD
- ▶ Full compatibility between input and output files. Only nodal coordinates are modified

ParaMesh Target Geometry

- ▶ Import Options
 - IGES
 - STEP
 - TETIN

Coupling

The output of ParaMesh or ParaBatch can be combined with any existing optimization products (ANSYS Optimization, ANSYS PDS, BOSS Quattro, modeFRONTIER, Isight, Optimus and SOL200, ...)

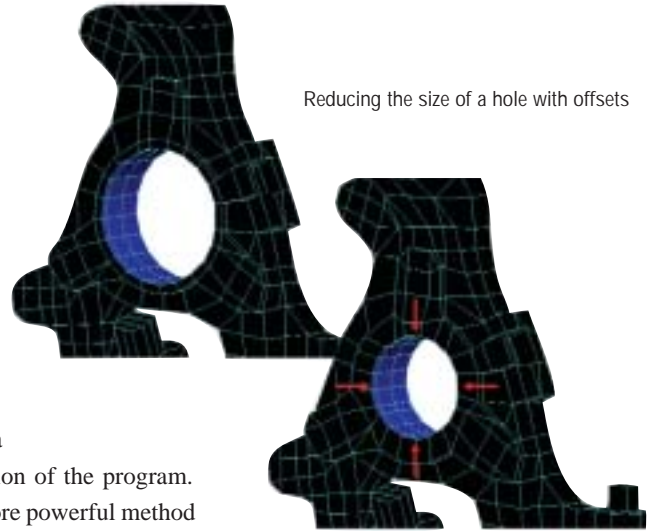
Customization

ANSYS, Inc. will adapt ParaMesh to a specific case (custom input/output, special functions...)

export deformed meshes from either a ParaMesh or a DesignXplorer VT database.

ParaBatch can export deformed meshes quickly and automatically without any user intervention, which allows ParaBatch to be utilized in two different ways.

The first one allows exporting a single mesh from a single execution of the program. The second one — which is the more powerful method — is to create multiple deformed meshes. This series of meshes can be used for DFSS studies, optimization, sensitivity analyses and what ifs! ParaBatch can be integrated into an automatic optimization loop or coupled with commercial software such as Insight, Optimus, modeFRONTIER and other optimization software making a powerful combination.



Reducing the size of a hole with offsets

ParaMesh Target Geometry Module

What do you do if your CAD model and your analysis model are not in sync? ParaMesh allows models that have gotten out of sync with the CAD model to be quickly and easily modified to match the current CAD geometry. The target geometry capability leverages both the work done by the analyst in setting up the analysis model as well as the work done by the CAD designer

in refining the CAD model. Bringing both of these

into agreement means providing the best possible answer for decision making, and with ParaMesh, this is accomplished in a very efficient manner. Also, the new target geometry capability offers an unlimited capacity for complex modifications to existing models. For example, if an automobile trunk has been redesigned for better aerodynamics, ParaMesh can be used to morph a portion of the structural analysis model to the new CAD target geometry while leaving the other portion unchanged.



Target geometry piston example

Find out how ANSYS ParaMesh can streamline your process and make your “Time to Best Design” faster than your competition. Visit www.ansys.com for more information.