

ASSEMBLY ANALYSIS:

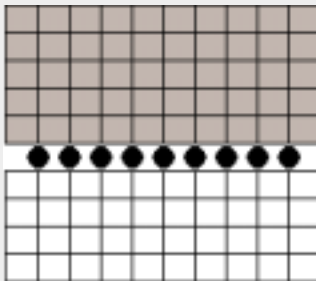
CONSIDERING TECHNIQUES
FOR ACCURACY

Assembly analysis is one of the most complex techniques for accurately predicting how a multi-component structure will perform. Through the automation of proven techniques, engineers can realize the benefits gained by properly performing assembly analysis. However, the user needs to understand the capabilities, limitations, and risks of performing assembly analysis. This discussion reviews the most common techniques used by computer-aided engineering software developers.

The Common Mesh Technique

The most basic form of assembly analysis is the “common mesh” technique for simulating “perfectly bonded” geometry. This finite element analysis technique takes assembly geometry and produces one continuous finite element mesh throughout the structure, assigning different material properties to the different parts that comprise the assembly. The common mesh technique is appropriate for assemblies that should perform as though they are perfectly bonded. Examples of this include parts welded together (if weld strength is not an issue) or bonded materials such as metal plating. While this technique gives reasonable answers for such situations, users need to be aware of limitations to the common mesh method. Because the common mesh technique becomes unwieldy when meshing elaborate assemblies, this method has little merit for the analysis of real-world assemblies.

The common mesh assembly analysis technique uses one continuous finite element mesh throughout the entire assembly, turning it into one big part with multiple material properties. Accuracy for individual parts is forfeited to maintain a constant mesh.



The node-to-node technique requires knowledge of where parts will contact prior to analysis. This technique does not allow parts to move based on real-world conditions.

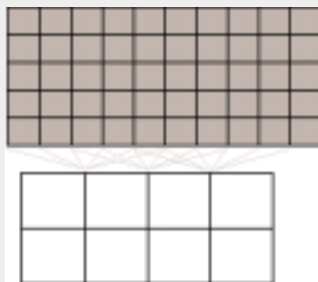
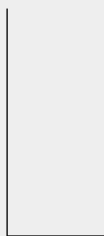
The node-to-node technique requires knowledge of where parts will contact prior to analysis. This technique does not allow parts to move based on real-world conditions.

Real-world assemblies typically have parts that move. The common mesh method assumes that all components are rigidly fixed to each other. There is no way to account for any interaction among parts because the finite element mesh treats the entire assembly as one big part. There is no way to account for any nonlinear behavior associated with individual parts. In short, unless you assume the entire assembly is perfectly bonded, you will end up with misleading, if not blatantly wrong, answers. Typically, the common mesh technique indicates areas of stress or deformation that are inaccurate due to disregard for individual part movement.

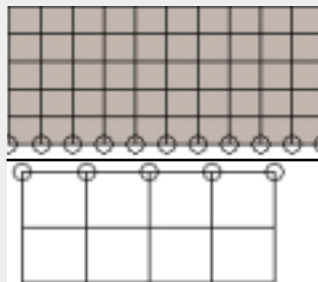
The Node-to-Node Technique

Moving up the assembly analysis food chain is “node-to-node” assembly analysis, also known as “gap” analysis. The primary difference between the common mesh technique and the node-to-node technique is the basic consideration of interaction between parts. Essentially, node-to-node assembly analysis simulates contact between parts with contact elements. Taking a common mesh, contact elements are placed in between the nodes that separate one material property of an assembly from another. This technique imposes strict requirements on mesh generation for the entire assembly, often at the expense of accuracy for any one part. In fact, mesh generation may not succeed at all; in case of many-component, real-world assemblies, common mesh technique platforms may refuse to mesh the part. This technique assumes that the accuracy needs of the whole assembly are greater than the needs for accuracy regarding an individual part, which is frequently not the case.

However, while some part movement is considered, there are serious limitations to this technique above and beyond the assembly mesh issue. To use node-to-node contact elements, you must know the location of contact beforehand. In multiple-part assemblies with individual part movements, this can be very difficult. To make matters worse, these types of contact problems usually allow for only relatively small sliding between the contact surfaces of parts (even in the case of geometric nonlinearities). So, even if you can accurately guess which areas of which parts may come into contact, you are limited to problems where the nodes of individual parts line up perfectly, relative part sliding is negligible, and deflections or rotations of parts remain small. Node-to-node

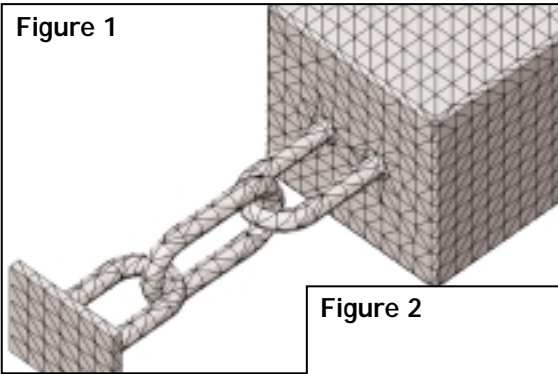


The node-to-surface technique allows parts to move realistically throughout the assembly. It also allows variable meshing of individual parts for part-specific results.



The surface-to-surface technique provides the best of all worlds. It can solve for rigid-to-flexible or flexible-to-flexible surface conditions. Meshes can be optimized for part-specific results.

Figure 1



In Figure 1, an assembly is meshed using a common mesh, which treats the entire assembly as one part with multiple material properties. With DesignSpace from ANSYS, Inc., you can treat components independently, meshing each only to the level of detail your analysis requires. In this case, the middle link, Figure 2, is the focus, so it is meshed for more detail than other components. Figure 3 shows the result: a targeted, robust analysis, indicating deformation and the areas of greatest stress (indicated by lighter-coloration).

Figure 2

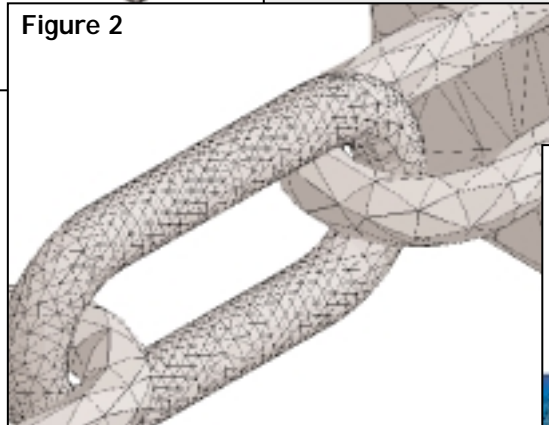
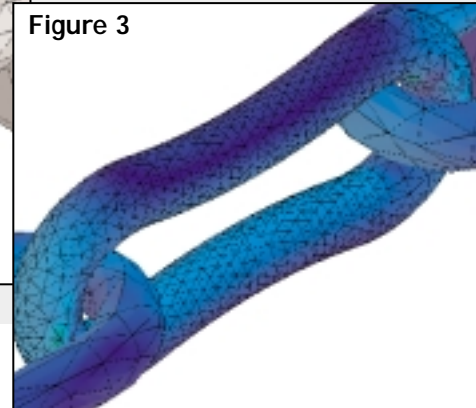


Figure 3



assembly analysis can be frustrating. Result convergence is typically cumbersome at best, and often insufficient.

The Node- and Surface-to-Surface Techniques

Programs for performing true assembly analysis employ advanced techniques. One such method is the "node-to-surface" technique. The primary difference between the node-to-node and node-to-surface techniques is the ability to account for large deformations and large relative part sliding, without requiring knowledge of the exact location of the contact areas beforehand. The node-to-surface technique also eliminates the need for compatible meshes across contacting parts. Node-to-surface elements are typically used to model point-to-surface contact, such as that found with snap-fits.

Another, more advanced method, is the "surface-to-surface" technique. The surface-to-surface technique accounts for real-world interaction between parts of an assembly. This technique uses either "rigid-to-flexible" or "flexible-to-flexible" contact elements. These contact elements use a target surface and a contact surface to form a contact pair. Once formed, the real-world interaction

among parts of an assembly can be simulated.

Conclusion

Many Assembly analysis problems are highly nonlinear and require significant computer resources to solve; they also present two significant difficulties. First, you generally don't know the areas of individual part contact until you have run the problem. Depending on the loads, material, boundary conditions, and other factors, individual part surfaces can come into and go out of contact with each other in a largely unpredictable manner. Additionally, assembly analysis programs should address multiphysical effects, such as the conductance of heat and electrical currents in the areas of part interaction or the effects of thermomechanical stresses on an assembly's natural resonant frequencies. Finally, many assembly analysis applications will require the ability to fine-tune meshing on a component basis. It is important that an application not only provide this functionality, but make it as easy to deploy as point-and-click, automating the discrimination of one component from others, reducing your time investment.

Before investing in software to perform real-world assembly analysis, take a look at the company that develops the underlying technology. Are they known for their non-linear analysis capabilities? Can they take multiphysical effects into consideration at once? Do they have clients that have used their software successfully to solve real-world assembly analysis problems? Before you start spending your money and (more importantly) your time on assembly analysis software, take the time to ask the experts. Consult with your company's professional analyst or contact a local engineering analysis consulting firm.

Typical Real-World Assembly Analysis Procedure

1. Create and mesh your assembly model comprised of individual part files. Beware of programs that join all parts together to produce one continuous mesh throughout the assembly. This usually indicates that the program uses inferior common mesh or node-to-node contact element techniques.
2. Identify interactions between parts of an assembly. Are large displacements being considered? Beware of programs that generate contact, or "gap" elements between specific nodes. They do not allow for real-world part movement.
3. Apply environmental conditions to your assembly, such as loads and supports. Look for programs that automate the application of "intelligent" loads and supports such as bolt loads and pin supports. You can get wrong results for an entire assembly analysis by applying a wrong load.
4. Solve and review results. Look for programs that allow you to query individual parts for individual results. This provides a quick visual check to make sure the results make sense. If you are getting counterintuitive stress distributions, it is probably due to problems associated with a common mesh technique.

ANSYS®

ANSYS, Inc.

Southpointe
275 Technology Drive
Canonsburg, PA 15317

ansysinfo@ansys.com

T 724.746.3304

F 724.514.9494

Toll Free USA and Canada:

1.800.WE.R.FEA.1

(1.800.937.3321)

Toll Free Mexico:

95.800.9373321

Regional Offices:**North America**

na.sales@ansys.com

T 724.514.3122

F 724.514.3115

International

int.sales@ansys.com

T 724.514.3086

F 724.514.3115

Europe

eu.sales@ansys.com

T 44.118.9880229

F 44.118.9880925

<http://www.ansys.com>

ANSYS and DesignSpace are registered in the U.S. Patent and Trademark Office. All other trademarks and registered trademarks are the property of their respective owners.

©1999 SAS IP, Inc.
All Rights Reserved.
Printed in U.S.A.
MP-P20-9/99