# Preliminary Design Studies using SimSolid

## Frequently Asked Questions

### Simulation Technology

1. **What differentiates SimSolid from other simulation solutions?**
   - SimSolid is a fully-featured structural analysis solution that can analyze complex parts and large assemblies efficiently on a desktop class computer. It eliminates geometric simplification and meshing, the two most time-consuming tasks in other simulation software. The solution is both fast and accurate and helps achieve meaningful design inputs in seconds to minutes without the need for high-performance computing machines.

2. **How is it different from traditional FEA (h and p-element)?**
   - SimSolid is a generalization and extension of the finite element method. In SimSolid, each part can be represented by a single region or large general shaped regions.
   - Solution refinement is performed via p-enrichment or via introduction of special non-polynomial functions. Solution adaptation is based on relative local energy density changes and absolute errors on region boundaries.
   - The degrees of freedom are also not point wise, they can be associated with volumes, surfaces, lines and/or point clouds. DOF are integrals over corresponding geometrical objects, not nodal. Depending on the solution adaptation, there can be many degrees of freedom for a single associated geometry object.
   - Boundary compatibility is met approximately between regions and is adjusted during each solution pass.

3. **Is it based on iso-geometric analysis?**
   - SimSolid is not based on iso-geometric analysis. Unlike SimSolid, IGA is not mesh free. It is a finite element method that requires analysis-ready geometry and hence (iso-geometric) elements.

4. **What makes SimSolid so fast?**
   - The number of degrees of freedom used to represent the geometry can be orders of magnitude less compared to traditional FE. SimSolid is also multi-threaded.

5. **Is it a displacement or force-based formulation?**
   - It is a displacement-based finite element formulation.

6. **Is SimSolid accurate?**
   - The accuracy of SimSolid across a range of typical solution domains has been confirmed by comparing against well-known reference solutions. Please refer to the validation manual.

7. **What are the error criteria used?**
   - There are two primary absolute error criteria used: (1) displacement error at boundary with prescribed displacements, (2) traction error at boundary with loads. There are several relative error criteria (frequency, strain energy, etc.) used.

## Capabilities

1. **Does it leverage GPUs?**
   - Today, SimSolid only runs on CPUs and does not leverage any GPUs.

2. **Is there a limit on the number of parts or facets that SimSolid can handle?**
   - There is no limitation on the size of assembly that SimSolid can handle, it is only limited by the machine resources that are available.

3. **Why are the bolts in my assembly not recognized by SimSolid?**
   - Bolts are recognized by certain (hexagonal) head and shaft geometric patterns. If your bolt is not recognized, send us the geometry and we will see if we can add it.

4. **Does SimSolid only plot surface results?**
   - SimSolid calculates full 3D result functions. Currently, SimSolid result functions are evaluated on part surfaces by the means of a response surface mesh. Continuum results along the cutting planes will be supported in the future.

5. **Is SimSolid capable to solve path dependent problems with multiple nonlinear steps?**
   - All the solutions run in SimSolid are independent of each other. Multi-step nonlinear analysis is currently under development.

6. **Does SimSolid support composite materials?**
   - SimSolid currently only supports isotropic materials. Other material models including composite materials will be supported in the future.

7. **Does SimSolid support nonlinear contacts?**
   - Yes, SimSolid does support nonlinear contacts that can open and close. It currently only supports small sliding; large sliding is not yet supported.

8. **Does it support material plasticity?**
   - Yes, SimSolid uses deformable theory of plasticity to evaluate small strain plasticity in materials. You can define an elastoplastic behavior using stress-strain curve or K-n power law function.

## Best Practices

1. **Is it recommended to use STL files?**
   - STL should only be used for organic shapes such as lattices, topo-optimized shapes, etc. SimSolid does not import CAD surface or solid geometry, it instead uses a more efficient faceted geometric approach. Since STL files have a flat assembly tree structure, SimSolid determines the part face structure based on surface curvature that can lead to missing fillet faces. It is therefore recommended to use CAD geometries that have CAD part faces.

2. **What should be avoided in terms of poor geometry?**
   - The general rule is to use actual manufacturable geometry. It is also recommended to never merge parts.

3. **Is it possible to determine any geometric defects prior to running analysis?**
   - You can check any geometric defects such as overlaps/gaps, non-manifold faces etc. using the "check geometry defects" button in the Assembly workbench toolbar.

4. **Why does auto connections fail to connect parts in the assembly?**
   - Auto connections uses gap and penetration tolerances to connect parts in the assembly. Best practice is to set these tolerances just large enough to connect. It is recommended to keep connections with large overlaps away from areas of performance interest.

5. **How to choose contact refinement?**
   - The default normal contact resolution is adequate for most situations. Only increase the resolution for areas with detailed geometric requirements such as thin edges or where small and large parts are connected.

6. **How to choose master and mate part in a connection?**
   - The selection of master and mate part is arbitrary in SimSolid. The selection does not influence the results.

7. **How to ensure accurate stress results?**
   - The recommended workflow is to activate the checkbox 'adapt to features' under solution settings and use 4, 5 (or max. 6) solution passes. Use 'adapt to thin solids' only for thin curved geometry and it is recommended to apply locally using part groups.
   - Note that all solution adaption controls are either global or part based. There are no fine-grained (local region) stress controls at this time.

8. **How do I fix the error "unable to solve equation systems due to numerical instability?**
   - This error is typically due to either geometry or connectivity problems. Please make sure the connections have properly been created and there are no geometric defects in the model.

9. **What is the recommended computer configuration?**
   - SimSolid is only supported on windows platform. It is recommended to use a Windows 10 OS on an Intel I7 based computer with at least 4-cores and 16 GB of memory.