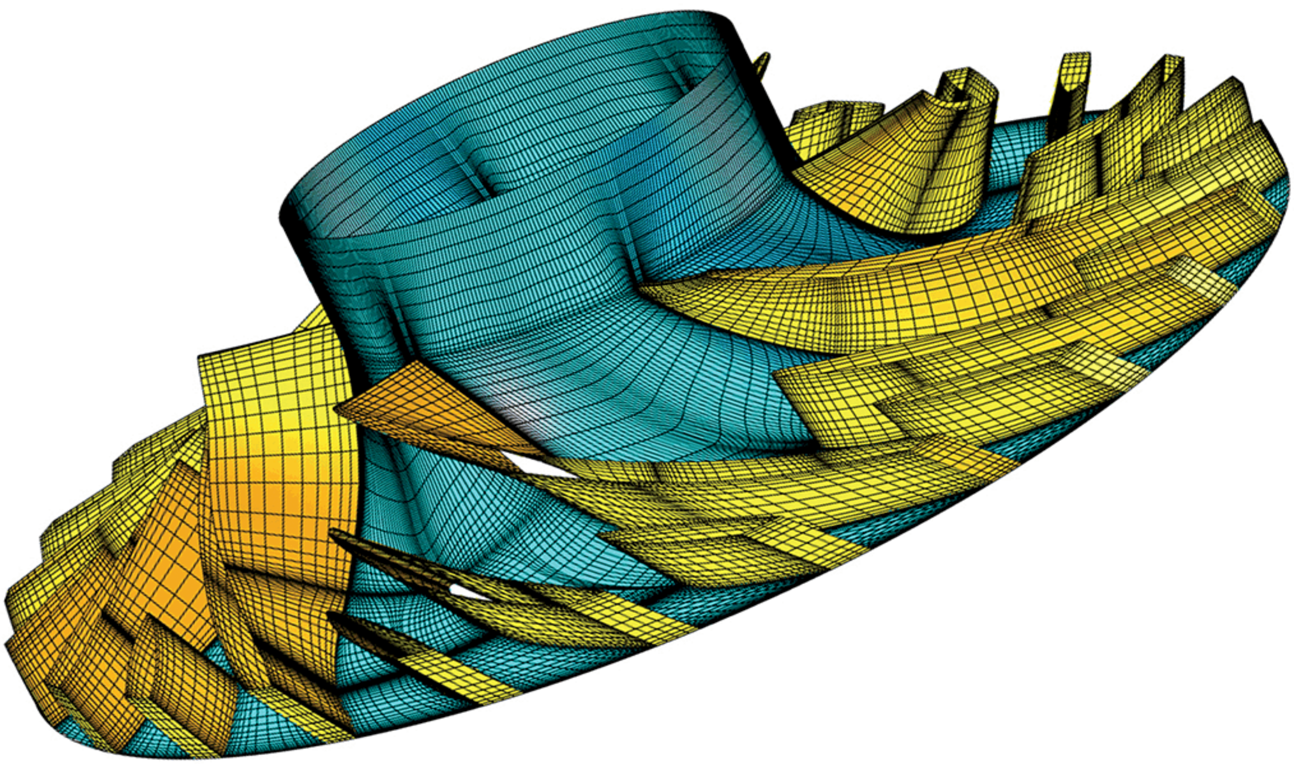
TrueGrid®

by XYZ Scientific Applications, Inc.



Mesh Generator and Pre-Processor for FEA and CFD Analysis

The True*Grid*® Advantage

True*Grid*® software is advanced meshing technology for FEA and CFD simulation codes. **True*Grid*®**'s unique approach to mesh generation saves time and produces quality, near orthogonal elements. Even complex models are meshed to superior quality.

The Projection Method

True*Grid*® meshes are formed more quickly and easily using the projection method. The projection method takes the menial tasks out of capturing geometric features in the mesh.

Multi-Block Structures

True*Grid*® uses a multi-block approach to mesh generation. Block structured hex meshes produce the highest quality meshes, ensuring accurate results. Multi-block structures minimize the effort of constructing complex models.

No Geometry Clean-Up

True*Grid*® faithfully imports CAD/CAM and solids model surfaces and there is no need for geometry clean-up. You can import your CAD/CAM or Solids Model in the standard IGES format and save time and frustration because your IGES surfaces will be flawless.

Parametric and Scripting Capabilities

True*Grid*® is both an interactive and batch mesh generator. In batch mode you can edit the scripting file to produce a parametric mesh. Save time by creating one quality parametric model to produce many variations.

Pre-Processing

True*Grid*® is a complete preprocessor for the simulation codes that it supports. **True*Grid*®** will write the entire input deck for a simulation code. The pre-processing tool in **True*Grid*®** makes it possible to take advantage of the advanced features available in the simulation code.

Geometry Library

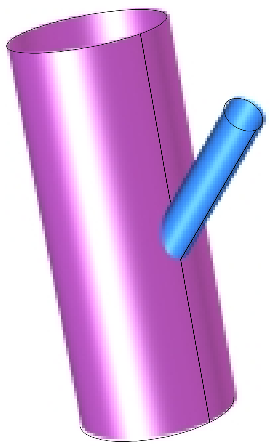
True*Grid*® has a built in geometry library so users can create their own geometry or add surfaces to geometry already imported from other files.

The Projection Method

True*Grid*® meshes are shaped by projecting faces of the mesh onto surfaces. This is known as the projection method. The projection method makes it possible for **True*Grid*®** to faithfully conform the mesh to any geometry.

The Projection Method is Quick and Easy

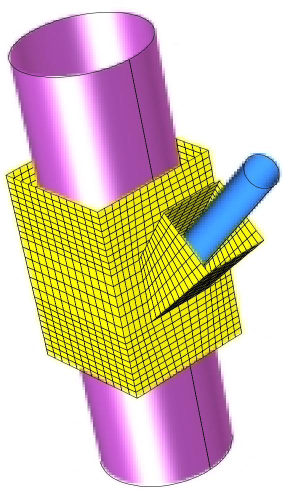
To use the projection method, specify the region of the mesh, select the surface to project to, then click on the Project button - and that's it. The projection method makes this Intersecting Pipes example easy.



Step 1. Create the Geometry

The first step is to define the geometry or import it from a CAD/CAM system or polygon data file. These surfaces were created in **True*Grid*®** using two `sd` commands:

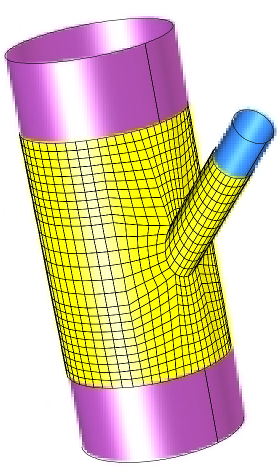
```
sd 1 cy 0 0 0 0 0 1 2
sd 2 cy 0 0 -2 .5 0 .5 .5
```



Step 2. Build the Topology

Next, a crude approximation of the mesh is constructed. First, the `block` command is used to build the topology. Next, `dei` commands are used to delete unneeded regions. Then, the `tr` command is used to move the mesh into place. Finally, the `res` command is used to force equal spacing of the nodes along the edges.

```
block -1 -13 25;-1 -6 -12 -17;1 -11 -17 27;
     -2 2 6 -2 -1 1 2 -3 -1 1 3
dei 2 3 ; -1 0 -4;;
dei 1 2; -2 0 -3;;
dei 2 3; -2 0 -3; 1 2 0 3 4;
dei 1 2 ;; -2 0 -3;
dei 2 3; 1 2 0 3 4; -2 0 -3;
tr 3 1 1 3 4 4 ry -45 mz -2;
res 2 1 4 2 4 4 j 1
res 2 1 1 2 4 1 j 1
res 1 1 1 1 4 4 j 1
```



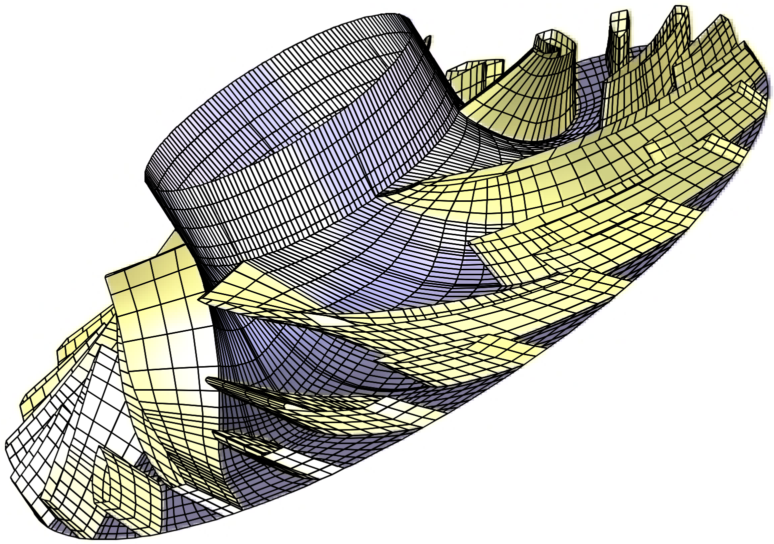
Step 3. Project the Mesh

Project the faces of the mesh to the appropriate surface of the geometry. This requires 2 commands:

```
sfi 2 3 ;-2 -3;-2  -3; sd 2
sfi -1 -2;-1 -4;;sd 1
```

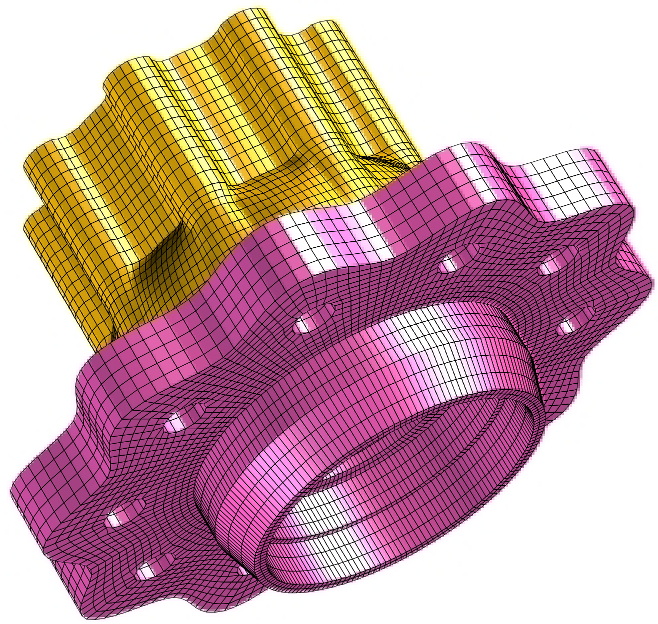
Notice that the edges of the mesh that are common to both pipes are automatically placed at the intersection.

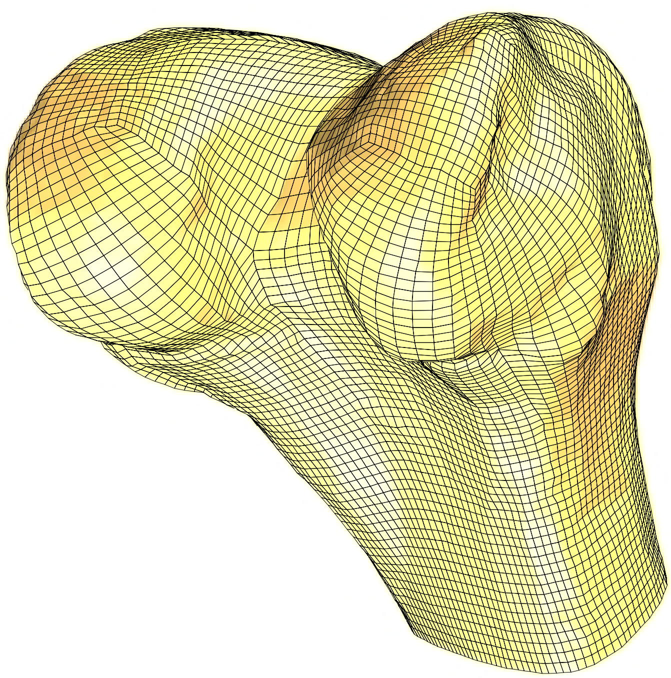
True*Grid*® Versatility



Fluid Applications
The projection method can be used to construct boundary layers for fluid flow around objects. The faces of the fluid around this impeller are shown to illustrate the nodal distribution of the fluid mesh.

Structural Applications
The projection method works great for assembling complex structures. This hub was formed using two parts. Each part was formed by projecting a small portion of the mesh and then replicating those regions to form the entire part. The number of replications for each part was different so careful planning was needed to make sure that the nodes at the interface between the two parts matched perfectly.





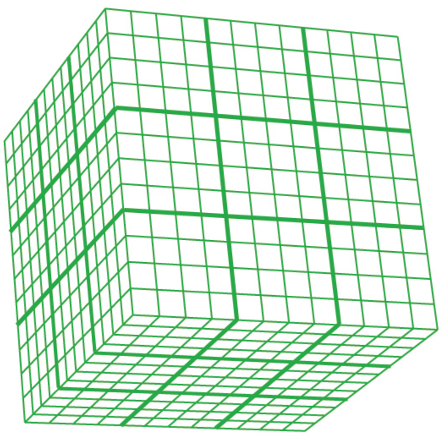
Biomechanical Applications
The projection method ensures that small features in biomechanical data are captured in the details of the mesh. The geometry for this bone was extracted from a CAT scan. The data was imported to **True*Grid*®** and the mesh projected to polygon surfaces. Elliptic smoothing algorithms were applied to give a smooth mesh while faithfully preserving the shape of the femur.

Multi-block Structure

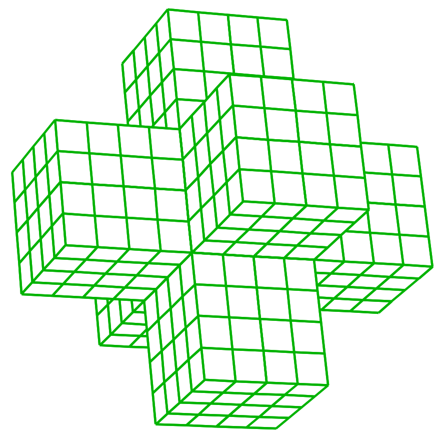
True*Grid*® supports multi-block structured meshes. Multi-block means that the block topology can be made from multiply connected blocks. Each block is composed of 3D hexahedral or 2D quadrilateral linear or quadratic elements arranged in rows, columns, and layers. These blocks can be moved, deleted, replicated, or glued to other blocks or parts.

Multi-blocks Save Time

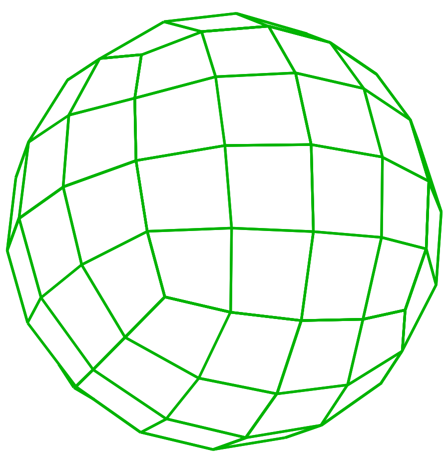
The main advantage of the multi-block structured mesh is efficiency. Blocks start out pre-connected. When a vertex, edge, or face common to several blocks is moved or shaped, you only have to take the appropriate steps once. If you were using single block technology you would have to take these steps once for every block that shares the same vertex, edge, or face. This is evident in the construction of a sphere. A multi-block part is used to design a "butterfly" topology to quickly achieve the best quality design for this problem.



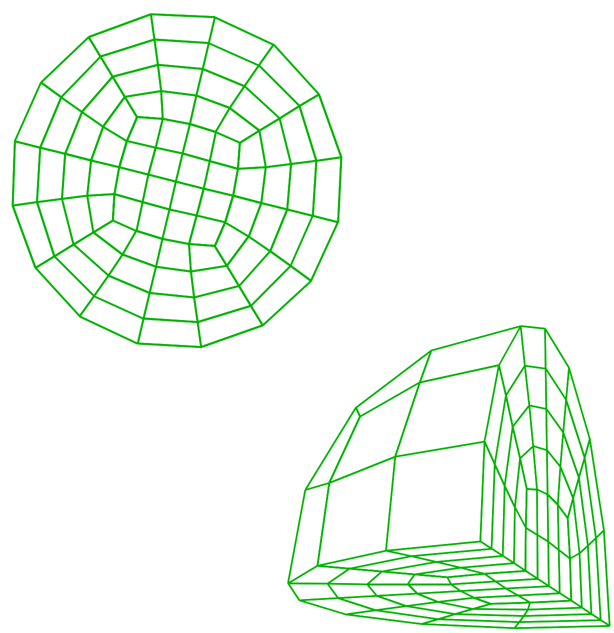
A 27 block multi-block part is created.



The corner blocks are deleted.



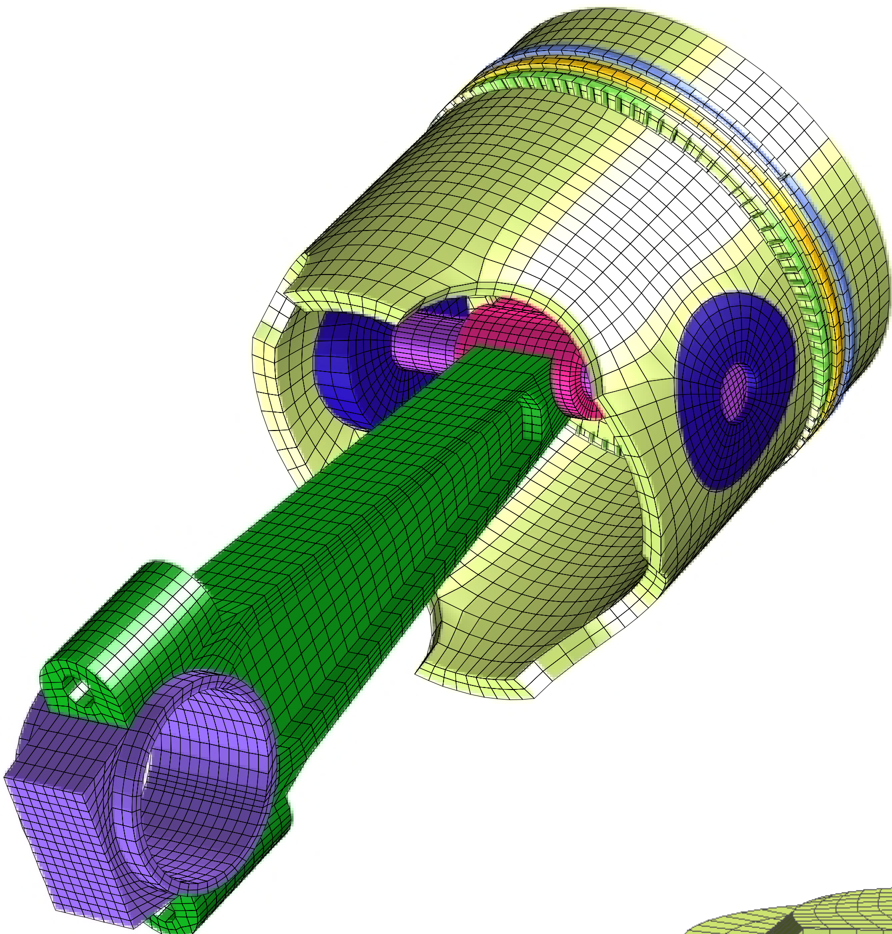
The faces of the mesh are projected to the surface of a sphere and smoothing algorithms used to evenly distribute the nodes.



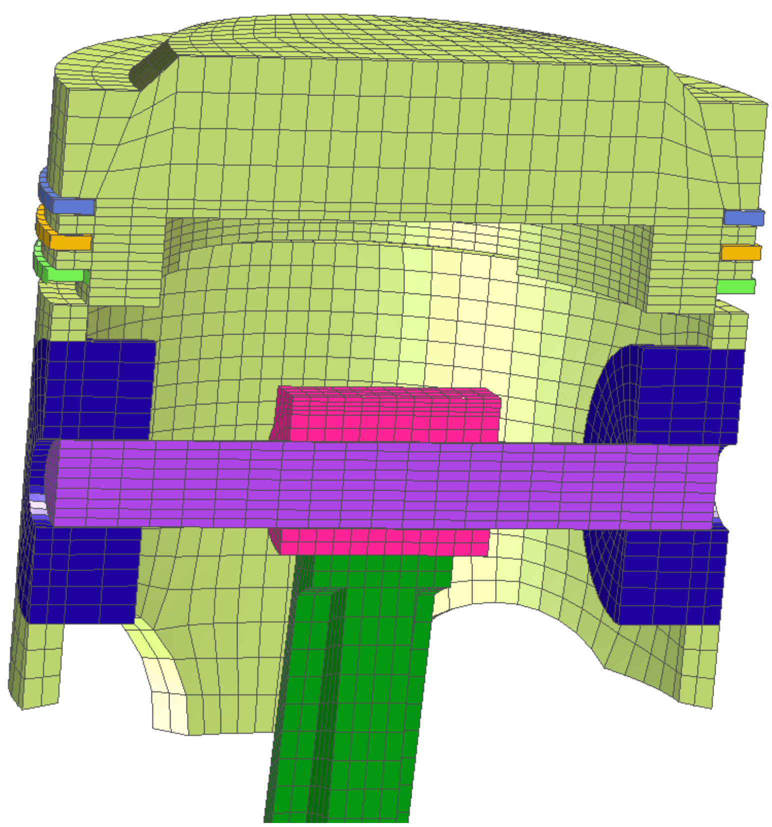
This sphere was formed in four easy steps in **True*Grid*®**. One half and one quarter view of the sphere shows the quality design.

Quality is Maintained Throughout the Design

Large models are easier to manage using multi-block structured parts. Users have control over the design of the topology and can plan for assemblies of many parts. This approach also ensures that quality hex elements are maintained throughout the model.



Piston and Rod
This piston and rod was easy to create because of the multi-block structured approach. This model has 9 parts and some of the parts where formed from many blocks.



Piston and Rod Interior
By slicing the model in half, it is clear that quality is maintained throughout. This would not be possible without using multi-block structured parts.

Parametric and Scripting Capabilities

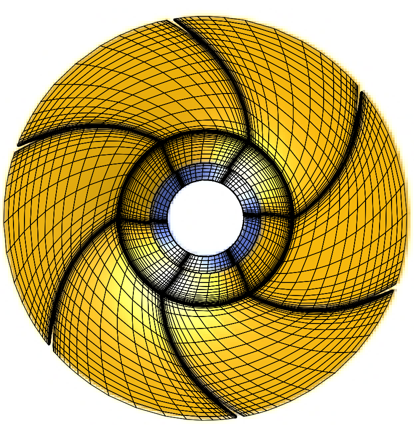
True*Grid*® is both an interactive and batch mesh generator. A scripting, or session file, is automatically generated during an interactive session. This scripting file can be edited so arguments to any command can be substituted with a parametric expression and the session file becomes a template to automatically generate meshes for an entire class of problems.

Parametric Features Include:

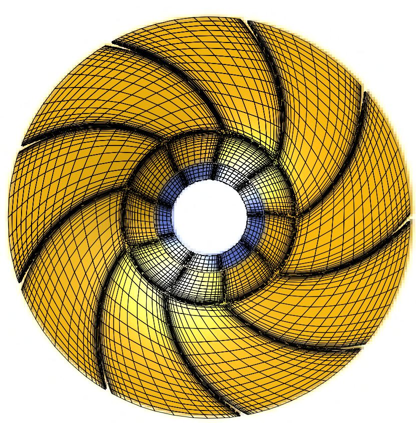
- Variables and arrays
- Algebraic expressions
- The creation of parametric curves and surfaces
- The definition and use of functions
- Global mesh density calculations
- Local mesh density calculations
- Inclusion of sub-meshes using the include command
- Conditional execution of commands
- Repetition of a group of commands in a loop
- The substitution of CAD geometry
- The if statement
- Iterative functions
- User defined functions
- Built-in **True*Grid*®** construction functions

Redesigned Features with Parameters

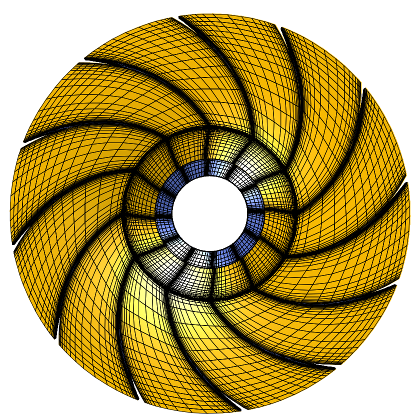
Parameters can be used instead of numbers for specific design features. For this impeller model, one of the parameter values designates the number of blades in the mesh. It took only seconds to change the parameter and generate a new model.



6 blade impeller



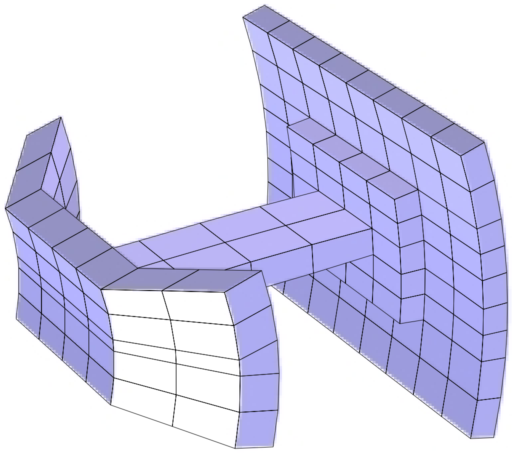
9 blade impeller



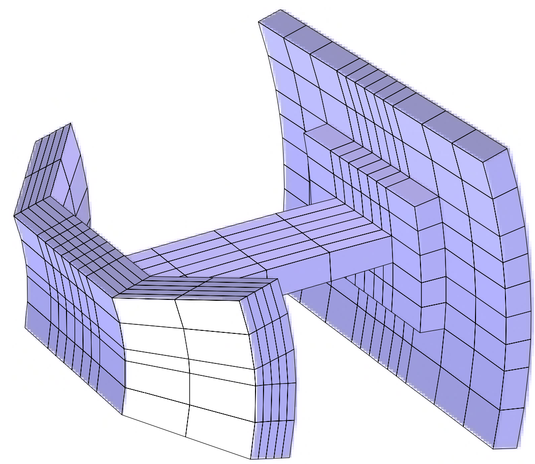
12 blade impeller

Use Parameters to Change Mesh Density

You can edit the block or cylinder commands in the scripting file to make changes to your mesh. The node numbers in the cylinder command are modified to change the mesh density in one region of this strut.



```
cylinder
1 2 6 7 8;
1 3 5 6 8 10;
1 3 4 6 7 9;
```



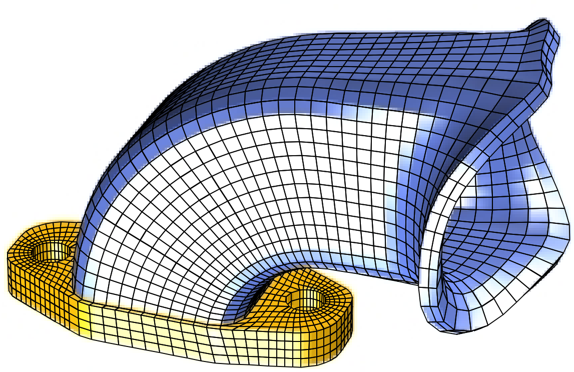
```
cylinder
1 6 10 11 12;
1 3 5 6 8 10;
1 3 4 10 11 13;
```

Use Parameters to Change the Design of a Model

Parameters can be used for specific design variables to easily redesign a model. The length of this collector inlet model is quickly changed with two parameters.

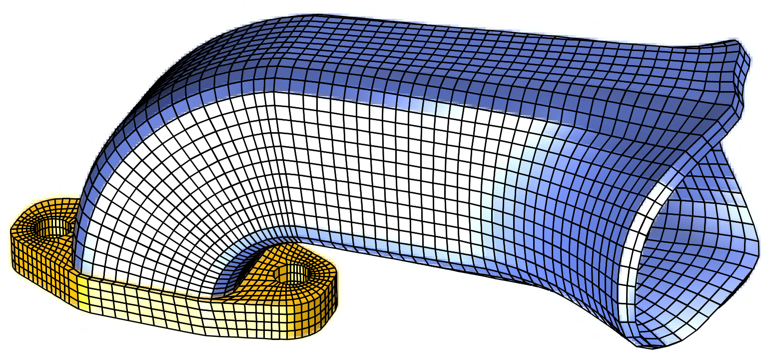
The parameters for this change are found on the following lines:

```
lmseq j 3 %nn
dom 1 2 1 5 4 2
x=x+%sf*(j-2)
```



Parameters effecting length set to zero

```
parameter
nn 0      c number of nodes
sf 0;     c scale factor
```



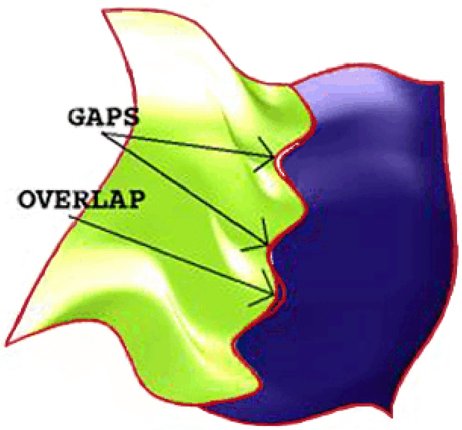
Parameters effecting length increased

```
parameter
nn 20     c number of nodes
sf 2;     c scale factor
```

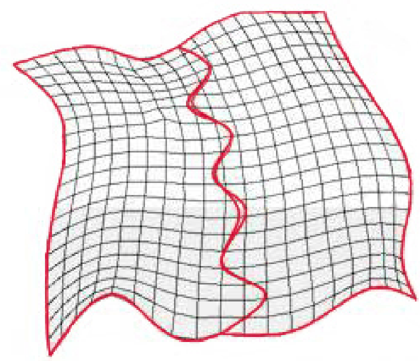
No Geometry Clean-Up

Every CAD system and solids modeler has a tolerance when the surfaces are written to an IGES file. This can cause surfaces to have gaps or overlaps. With **True*Grid*®** no time is spent perfecting geometry imported from a CAD/CAM or solids modeler because the projection method automatically ignores surface boundaries, gaps, and overlaps, and finds the intersection of surfaces even when they do not quite meet.

Create a Smooth Mesh Across Many Surfaces

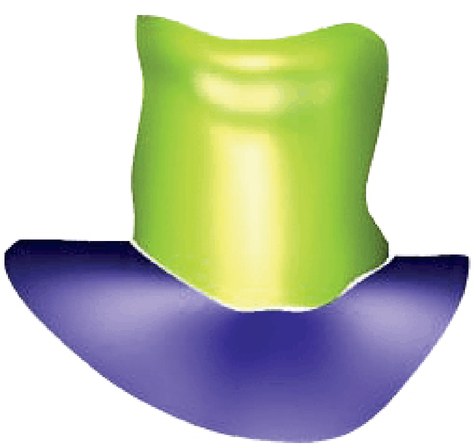


This geometry is representative of a typical IGES file with geometry composed of several surfaces that may have gaps and overlaps.

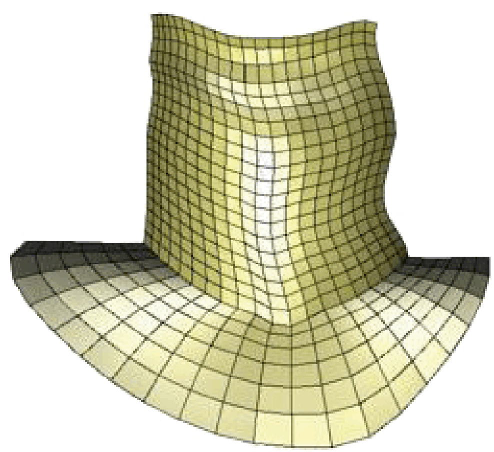


The surfaces are combined into one surface and the projection ignores the surface boundaries, gaps, and overlaps to create a smooth mesh across the geometry.

Automatically Place Mesh Lines at the Intersection of Surfaces



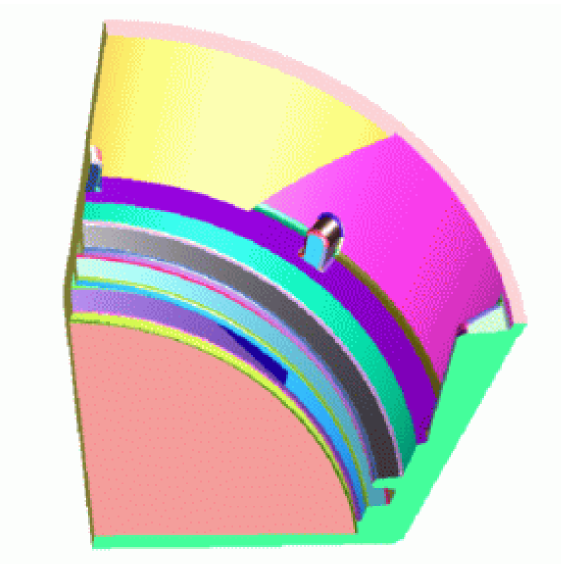
This geometry is composed of two different surfaces that do not meet. A mesh line is required at the intersection of the surfaces.



The projection method automatically finds the intersection of these two surfaces and places boundary nodes along that intersection.

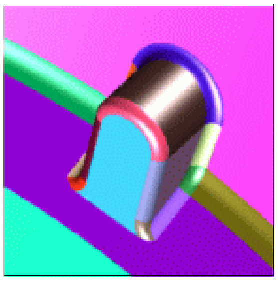
True*Grid*® **Combines Surfaces**

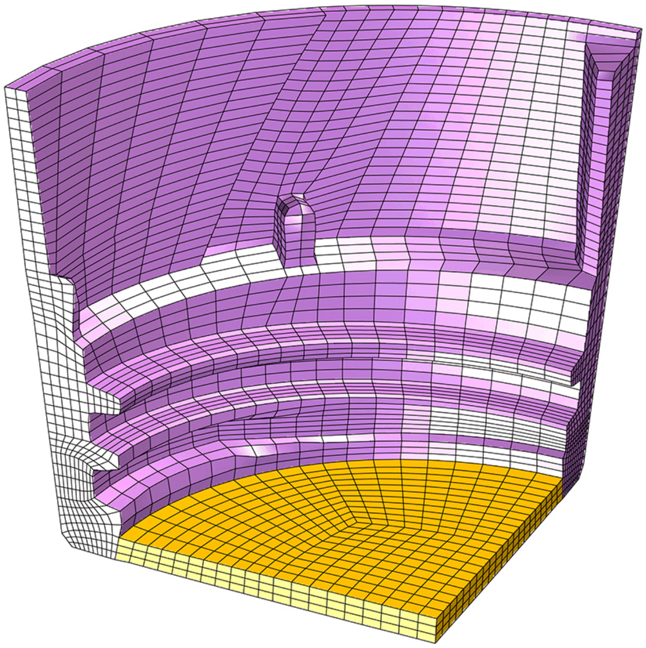
In addition to cleaning up the geometry, **True*Grid*®** also saves time by combining surfaces. Combining surfaces is desirable so that a single face of your mesh can cover many surfaces. Instead of building 100 meshes from 100 surfaces, form the union of the 100 surfaces in **True*Grid*®** (1 command) and project the mesh to this union (1 more command).



Cap Geometry with Many Surfaces
The helical thread in the cap is considered difficult to model with hex elements. Each color represents a different surface. Creating a mesh using each individual surface would be impossible.

Close-Up of Fillet
The filleted areas are commonly decomposed as shown. Just the single tab is composed of 16 surfaces. It is critical to be able to combine such surfaces into one to avoid spending too much time on a small feature.



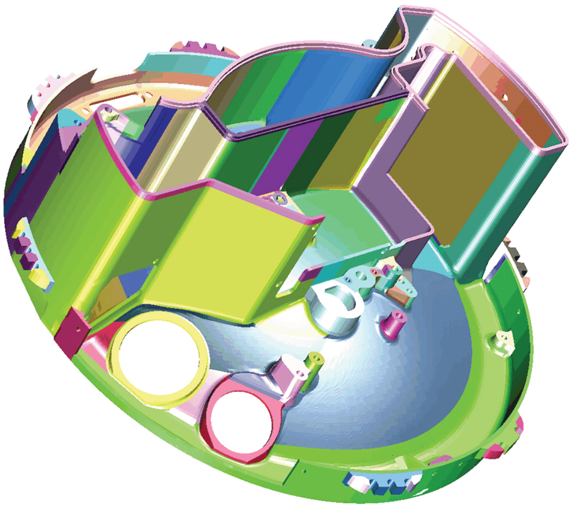


Easily Meshed with Combined Surfaces
In **True*Grid*®** a block mesh is projected to the combined surfaces and the problem is easily solved. The result is a quality mesh that was quick and easy to generate.

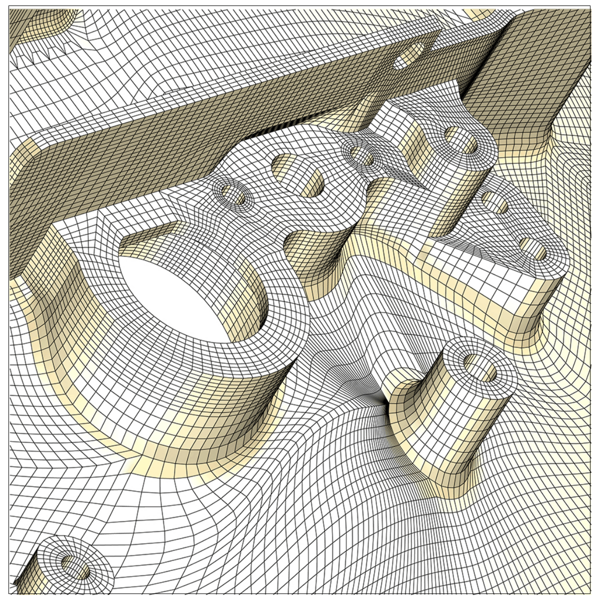
Assemblies

True*Grid*® 's assembly features helps to achieve a quality mesh for complex models. A face of one part can be glued to the face of another part using the block boundary interface command. Parts are easily replicated to form complex assemblies from simple components.

Glue Parts Together Using Block Boundaries

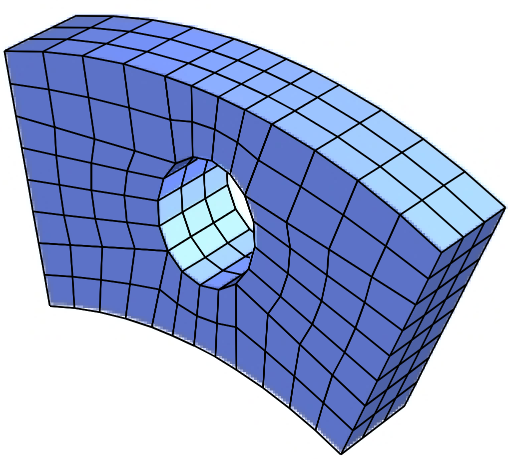


This complex model was broken into 177 parts then glued together to create a high quality mesh. Each color represents a different part.

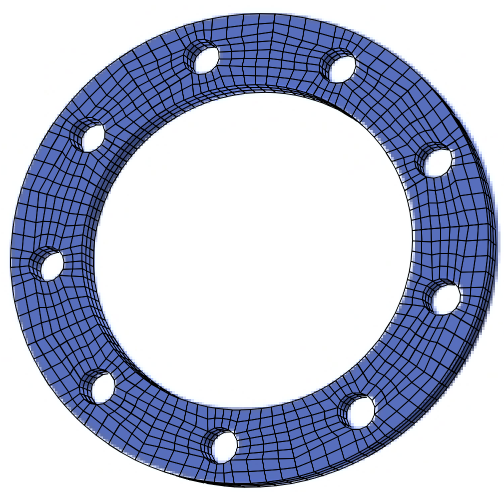


Zooming in on the mesh shows that the parts match perfectly.

Replicate Parts to Make Assemblies Easier



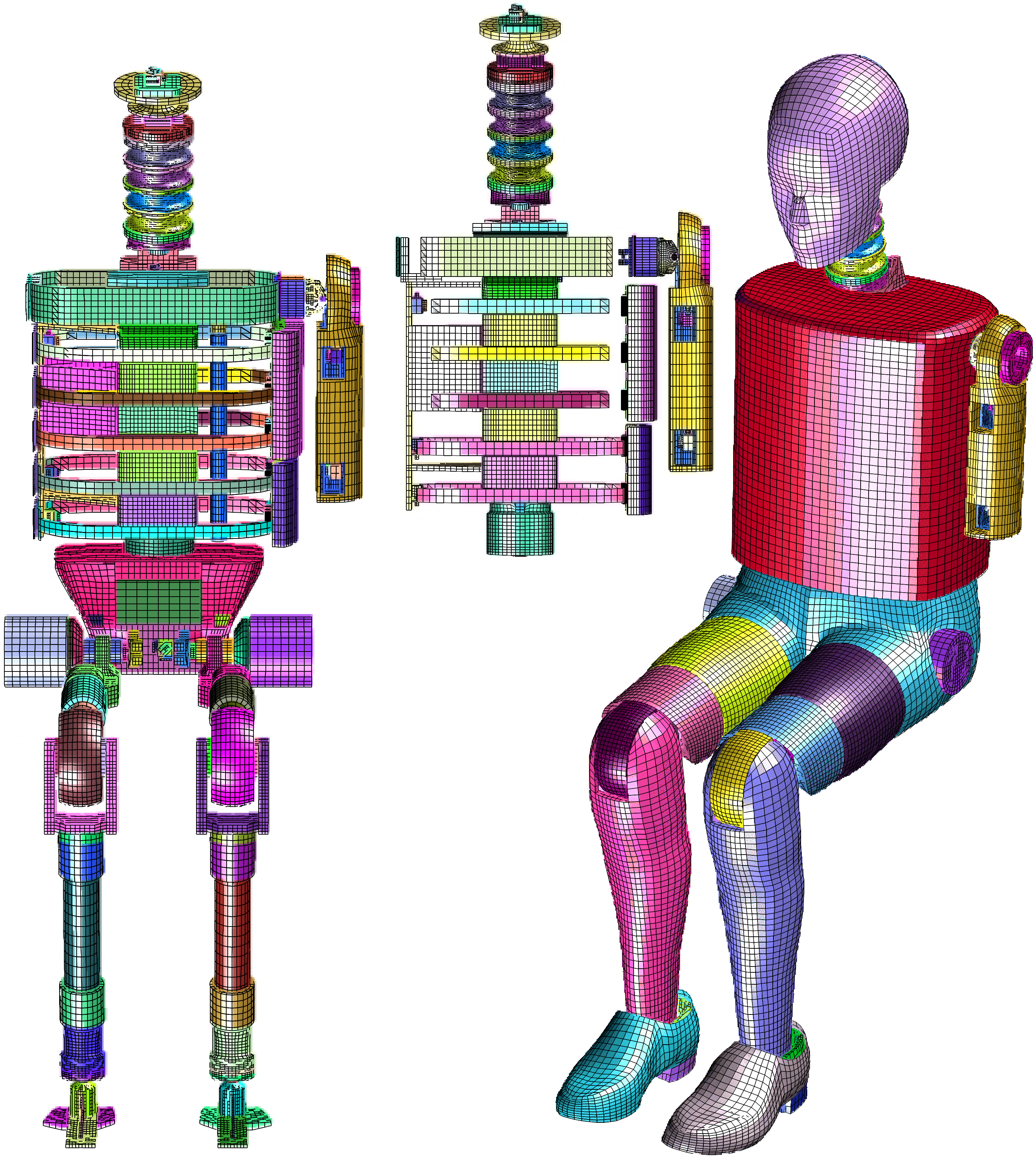
This part was created in **True*Grid***® using a multi-block structure of 9 blocks. The center was deleted and the faces projected.



The part was replicated using one command to form the entire ring.

Large Assemblies

The block boundary interface command provides the tools to assemble models with many parts and materials. This side impact dummy (SID) was created using several parts consisting of multiple layers. The mesh is based on NHTSA design drawings and outer surface scan data. The images below show the different layers used to construct the dummy and each color represents a different part.



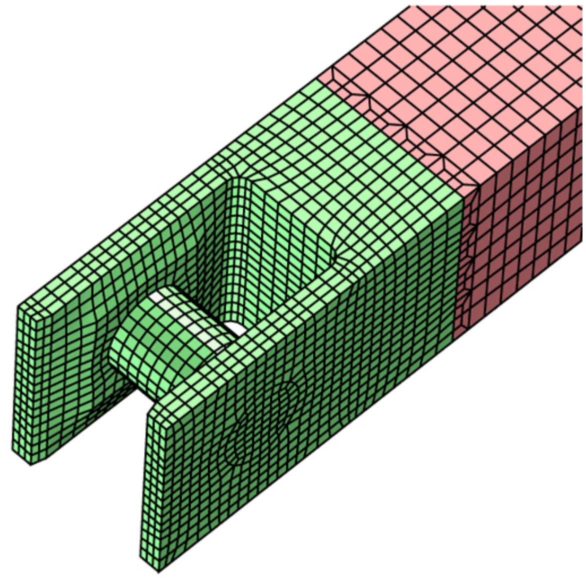
Crash test models provided courtesy of Livermore Software Technology Corporation. www.lstc.com

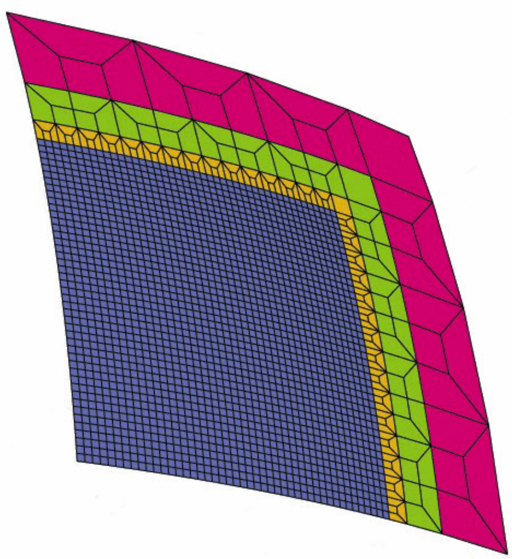
Transition Elements

The transitional block boundary command is a command for gluing parts with dissimilar element counts. **True*Grid***® can automatically transition hexahedral or quadrilateral meshes across block boundary interfaces using only hexahedral or quadrilateral elements, respectively. A row of hex (quad shell) elements at the interface are automatically replaced with a row that sews the two parts together to produce a node for node, edge for edge, and face for face matching across the interface.

Using Transitional Elements

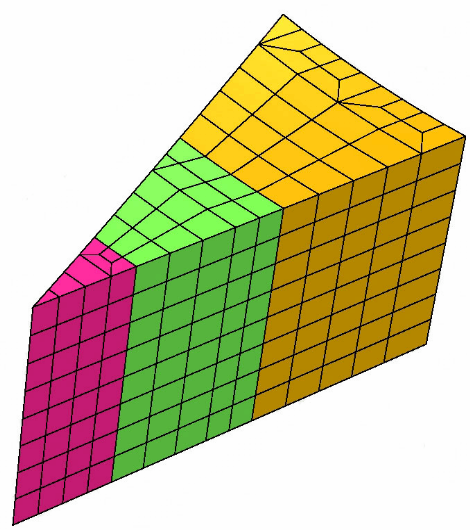
Transition Between Dissimilar Parts
If the number of elements in one part is 1:2 or 1:3 of the number of elements in another part, then a transitional block boundary is needed. The transition layer can be used to sew together parts with a different number of elements in both directions.





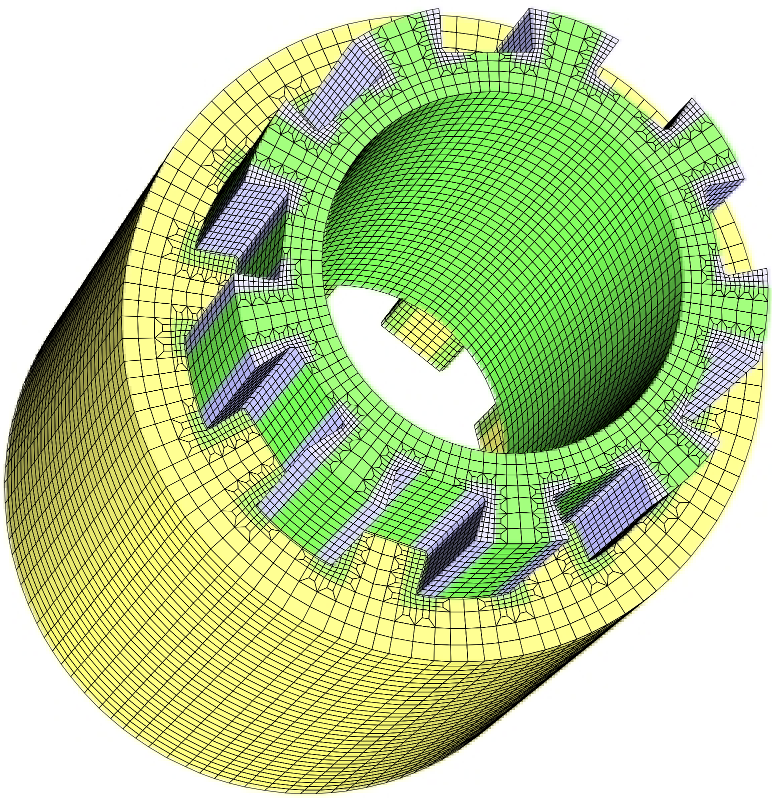
Transition Mesh Density
Transitional layers can be used to change mesh density. The mesh to the left has three transitions in two directions with a transition scale factor of 12 in each direction.

Use Transitions to Fill Irregular Regions
Transitions can be used with wedge elements to fill irregular regions where a complex solution may be needed.

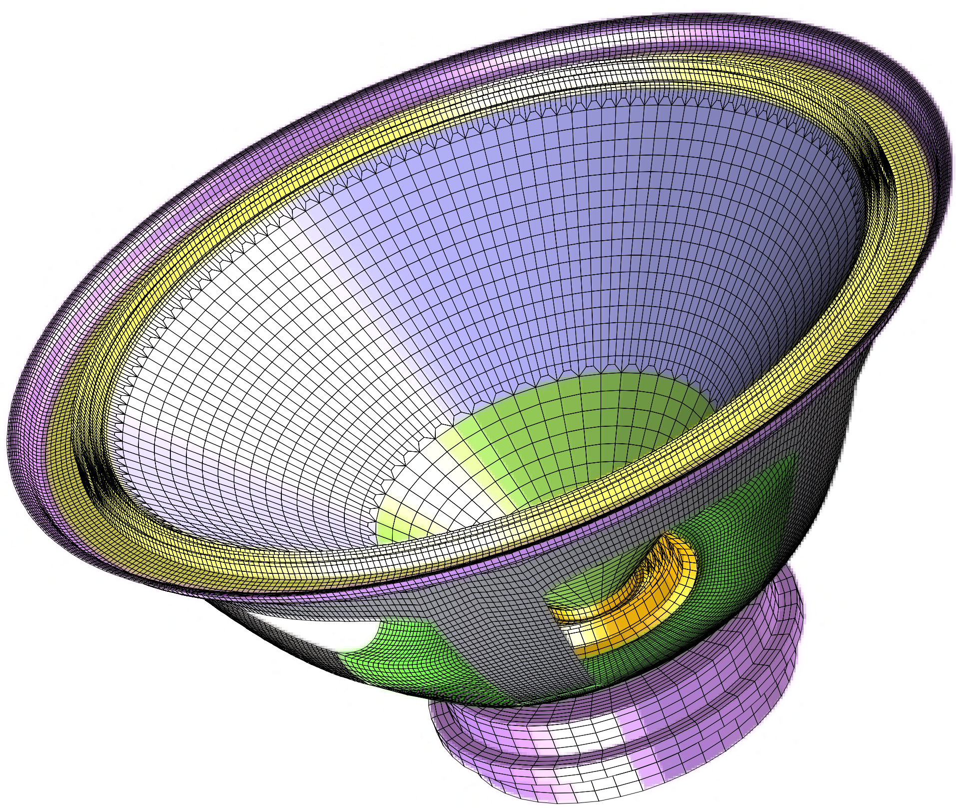


Transition Elements for Complex Models

Complex models are often not consistent in the number of elements from part to part. Transitional elements may be needed at every level to match parts of a complex model.



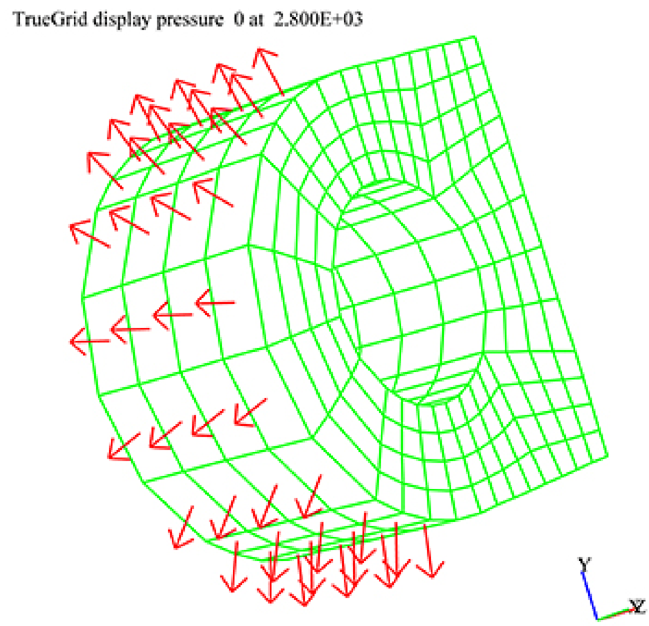
Spoke
This model has many different layers which require different mesh density. Transition element are used to match the parts.



Loud Speaker
The parts of this loud speaker were glued together with transitional elements that were automatically generated so that the interfaces between parts match perfectly.

Pre-Processing Capabilities

Loads and Conditions



- Initial and prescribed displacements
- Prescribed nodal rotations
- Prescribed nodal velocities
- Prescribed nodal accelerations
- Concentrated and distributed nodal loads
- Momentum deposition
- Pressure
- Bulk fluid
- Convection
- Thermal and electric prescribed flux
- Initial and prescribed temperature
- Radiation and radiation enclosure
- Heat generation
- Magnetic and electrostatic potential

Constraints and Boundary Conditions

With structures, a node can have a reduced set of degrees of freedom. With fluids, there are various boundary conditions such as an inlet, an outlet, or a wall, just to name a few. The list of possible boundary types is different for each fluids code.

Contacts and Interfaces

Some structural codes have a contact surface feature. In many cases, two opposing regions of the mesh must be identified. Also, these contact surfaces have characteristics that must be identified.

Properties and Element Types

Most simulation codes have a set of properties associated with each material model. These material model properties and the associated element types can be selected. Then elements are assigned a material model. This can be done by either selecting a region in the part phase or by assigning the material model to an element set.

Analysis Options

In order to run a simulation using a model from **True*Grid***®, one must first identify the analysis options and other global properties of the model. Many of these options can be selected within **True*Grid***® so that the final output file from **True*Grid***® can be imported to the simulation code, without modification, and run.

Geometry

True*Grid*® Geometry Library

Use the **True*Grid*®** Geometry Library to build your own geometry. Geometry surfaces include:

- Planes, spheres, cylinders, ellipsoids, cones, and tori
- 2D and 3D lines, polygonal curves, and arcs
- 2D and 3D cubic spline and NURBS curves
- 2D and 3D curves by algebraic functions
- 3D curves composed of edges of surfaces
- 3D curves from the intersection of surfaces
- Curves projected onto surfaces
- Concatenation of any set of 3D curves
- Curves of interpolation
- Lofted curves
- Surface edges and contours
- Surfaces of revolution
- Ruled, swept, and pipe surfaces
- Surfaces blended from 3 or 4 curves
- Cubic spline and NURBS surfaces
- Mesh and polygon surfaces including STL
- Combination of surfaces to form composites
- Surfaces by algebraic functions
- General transformations and normal offsets can be applied to the geometry

Import CAD/CAM and Solids Models

True*Grid*® imports trimmed surfaces and other geometry in the standard IGES format. Some of the CAD/CAM and solids modelers supported by the **True*Grid*®** IGES reader include:

- ANSYS
- CATIA
- CADDS
- NX
- ICEM DDN
- Pro/Engineer
- SolidEdge
- SolidWorks
- Surfcam CAD/CAM

Import Polygon Data

Import STL files and other polygon data files in ASCII or Binary format.

Import Existing Meshes from Several Formats Including:

- DYNA3D
- LS-DYNA
- MSC NASTRAN
- PATRAN Neutral File

Creating a Mesh in True*Grid*®

True*Grid*® gives users the tools to create quality hex meshes in a minimal amount of time. The example below illustrates the methods used in **True*Grid*®**'s to create a quality hex mesh with only 13 commands in 4 steps.

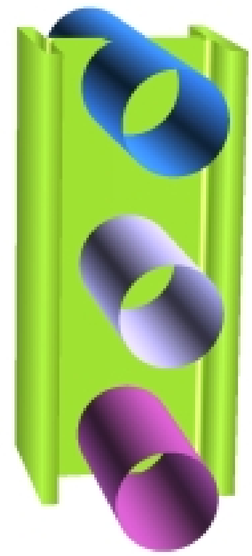
I-Beam Example

Step 1. Import or create the geometry

In this example we will create the surfaces from the **True*Grid*®** geometry library. The surfaces to the right were created in four commands

```
sd 1 cp 1;
sd 2 cy 6 0 0 0 0 1 3
sd 3 cy 6 -11 0 0 0 1 3
sd 4 cy 6 11 0 0 0 1 3
```

Where sd 2, 3, and 4 are the cylinders for the holes in the beam and sd 1 is the beam geometry that was created from an extruded 2D curve (also created in **True*Grid*®**).



Step 2. Construct the Mesh

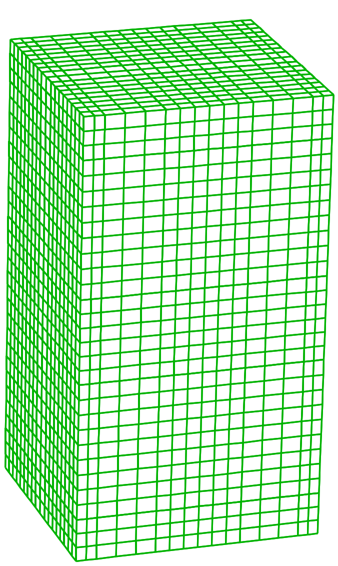
Next, create the multi-block structured part using the simple block command:

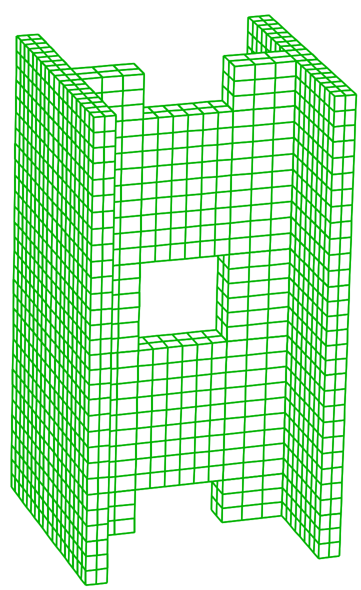
```
partmode i
block     2 3 6 3 2;
          3 9 6 9 3;
          8 2 8;
          0 1 3.9 8.1 11 12
```

Step 3. Delete Unneeded Regions

Regions are deleted to form the general topology of the mesh. This is accomplished with two dei commands:

```
dei 2 5;; 1 2 0 3 4;
dei 3 4; 1 2 0 3 4 0 5 6;;
```





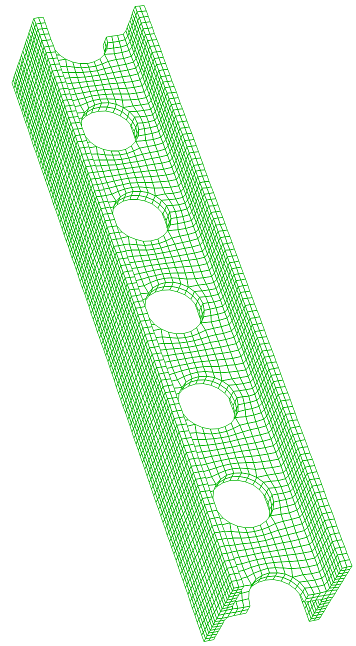
Step 4. Project the Mesh and Replicate

The faces of the mesh are projected to the surfaces using the following commands:

```
sfi -2 -5; 1 6; 1 -2 0 -3 4;sd 1
sfi -3 -4; -3 -4; 2 3;sd 2
sfi -3 -4; 1 -2; 2 3;sd 3
sfi -3 -4; -5 6; 2 3;sd 4
```

Next the mesh is smoothed and replicated to complete the model.

```
relaxi 2 5;; -2 0 -3;25 0 1
lct 2 my 22; my 44;lrep 0 1 2;
```

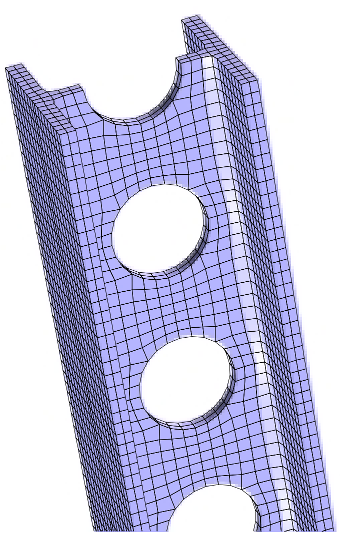


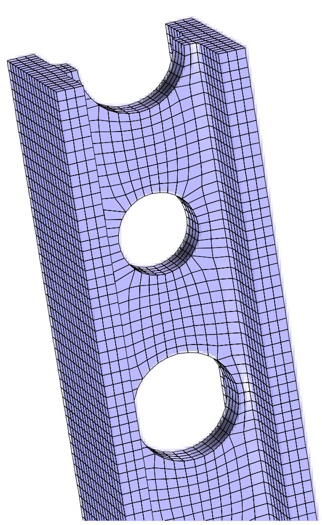
Step 5. Parameterize the Mesh

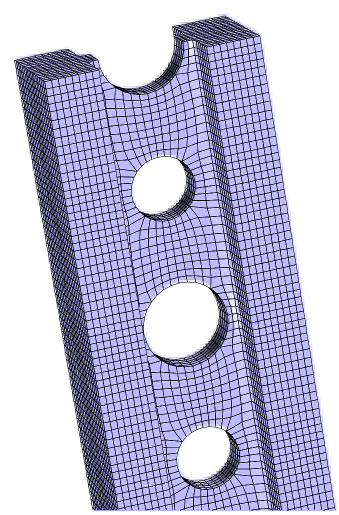
Although the I-Beam is complete, it may be necessary to make a parametric version in order to quickly run variations of the model. To do this the session file is modified with the following parameter statements. These parameters are integrated into the block command and any where else they are called.

```
para r1 .5       c  radius of the fillets
     th1 1.5     c  thickness of the web
     th2 1       c  thickness of the plates
     wb  10      c  width of the web
     pl1 11.5    c  width of the plates
     r2   3      c  radius of holes
     hg   5      c  gap between holes
      d   1.5 ;  c  mesh density
```

The following variations of the I-Beam were quickly generated by editing a parameter statement in the scripting file and rerunning.







True*Grid*® Export Options

Exports to FEA Simulation Codes

ABAQUS	ES3D	MARC	SAMI
ALE3D	EXODUS II	MSC NASTRAN	SAP 2000
ANSYS	GEMINI	NEi/NASTRAN	TOPAZ3D
AUTODYN	ICFEP	NIKE3D	
DYNA3D	LS-DYNA	PATRAN Neutral File	

Exports to CFD Simulation Codes

CF3D	FLUENT	PLOT3D at FSU
CFX4	GILA	PLOT3D at NASA
CFX5	GRIDGEN3D	STAR-CD
FIDAP	KIVA 4	TASCflow

True*Grid*® Availability

Platform	Operating System
Apple Mac Intel CPU	OSX
Apple Mac Motorola CPU	OSX
LINUX x86-32	Red Hat
LINUX x86-64	SUSE
LINUX Itanium 2	Red Hat, SUSE
UNIX HP PA-RISC	HP-UX
UNIX HP Itanium 2	HP-UX
UNIX SUN	SOLARIS
UNIX DEC ALPHA	TRUEUNIX
UNIX IBM	AIX
Windows x86-32	Windows 2000, XP, Vista
Windows x86-64	Windows 2000, XP, Vista

Licensing

Annual Licenses

An Annual License is a lease of the **True*Grid***® software for one year. The user has access to upgrades and technical support for the duration of the year.

Perpetual Licenses

A Perpetual License gives the right to use **True*Grid***® software forever. The user has the option to pay annual maintenance which gives access to upgrades and technical support for one year. This can be renewed every year.

Network Licenses

True*Grid*® can be run on any supported computer on your network. For each license purchased, one instance of **True*Grid***® can be run on the network. When multiple licenses of **True*Grid***® are purchased for a network, then that number of instances can be run simultaneously. Additional requests to run **True*Grid***® will be placed into a queue, and **True*Grid***® will run according to those requests as soon as an instance becomes available.

Trial Licenses

A fully functional, free Trial License of **True*Grid***® is available. The Trial License is valid for 60 days and comes with complete documentation and full technical support. Contact XYZ Scientific Applications to find out how to receive a Trial License.

Upgrades and Maintenance

Maintenance is sold on an annual basis for a Perpetual License in order to receive upgrades and technical support. You can download the newest version of **True*Grid***® over the internet, or request an upgrade DVD to be mailed to you.

Documentation

All documentation is available in electronic format. The ***TrueGrid***® *User's Manual*, ***TrueGrid***® *Examples Manual*, and ***TrueGrid***® *Tutorial* can all be accessed over the internet. These manuals contain over 1,000 colored diagrams and examples. Documentation is easily downloaded or you can request a hard copy to be mailed to you. If you would like more information about **True*Grid***®, you can contact XYZ to receive free colored brochure with detailed information.

Request a Quotation

To receive a quote or to purchase any of the licenses or documentation discussed above please contact us at:

XYZ Scientific Applications, Inc.
1324 Concannon Blvd.
Livermore, CA 94550
Phone:925-373-0628
Fax: 925-373-6326
www.truegrid.com
e-mail: info@truegrid.com

True*Grid*® by XYZ Scientific Applications, Inc.

About XYZ Scientific Applications, Inc.

XYZ Scientific Applications, Inc. is dedicated to developing advanced meshing tools for FEA and CFD analysis. XYZ began development of its flagship product, **True*Grid***®, in 1991 and continues to make technological advances to the software. In addition to developing **True*Grid***®, XYZ also uses **True*Grid***® for mesh generation consulting services.