Mesh Generation

Timothy J. Tautges
Principle Member Technical Staff
Sandia National Laboratories

Adjunct Professor, Engineering Physics
University of Wisconsin-Madison
What is mesh generation?

- Geometry from CAD (SDRC Ideas, Pro/Engineer, etc.)
- Mesh type determined by analysis; tetrahedra, hexahedra most common
What Kind of Mesh Should I Use?

- Tetrahedra
  - Automatic generation possible
  - Quadratic elements (10-node tets) typically used
  - More elements/nodes to resolve a given domain
  - More cpu time per unit accuracy *for some analyses, but not all!*

- Hexahedra
  - *Much* more difficult to generate (but getting better…)
  - Fewer elements/nodes
  - Conventional wisdom says they perform better (very little hard data to show that though)

- 2d plates/shells
  - All-quad mesh generation solved
  - Similar arguments regarding number elements, accuracy
  - ?
Overall We Use A Geometry-Focused Process

Why:
- Less interactive detail
- Fine geom $\approx$ Coarse geom
- Basis: initial CAD model

Model generation:
1. Start with well-defined design model
2. Use simplification strategies to obtain analysis model
3. Generate mesh

Geometry:
- “steel case”
- “impact”
- “foam”
- symm plane
- position

Mesh1:

Mesh2:

Material  Bound Cond  Init Cond

Sandia National Laboratories
Geometry Concepts

Basic Geometric Model:

- **Vertex**
- **Edge**
- **Face**
- **Volume**
  
  Non-manifold (shared) face

• Basis for most interaction with geometry in Cubit/Claro
  
  - Each geometric “entity” resolved by one or more mesh entities
    
    - Vertex---Node
    - Gedge----Edge
    - GFace----Face (Tri, Quad)
    - Volume----Element (Tet, Hex)

• Other geometry tools:
  
  - **Imprint**: “stamps” V/E/F onto neighbor
  
  - **Merging**: changes manifold to non-manifold model

- 2 unshared faces
- 1 shared face
Tetrahedral Mesh Generation

- **Automatic (Delaunay) meshing algorithm (almost) always used for volumes**
- Surface meshing using Delaunay (flat/parametric surfaces) or advancing front (curved/non-parametric surfaces)

- Boundary recovery
- Sliver removal (3D)
Tet Mesh Generation Process

Geometry Validity

Mesh Size/Interval Assignment

Size = .01

Mesh Scheme Assignment

Scheme = tetmesh

Mesh Generation

Mesh Quality Analysis
Hexahedral Mesh Generation

- No “automatic” hex meshing algorithm known!
- Lots of semi-automated algorithms
Hexahedral Mesh Generation

- No “automatic” hex meshing algorithm known!
- Lots of semi-automated algorithms

Quad (surface) algorithms
Sweeping (extrusion)
N-Side Primitives
Multisweep

Combined Use in Large Assemblies

- Decomposition
- Interval assignment

B61 Antenna Support
Structural Analysis
~258K Hex elements
Hex Mesh Generation Process

Toolkit Approach

1. Geometry Validity
2. Geometry Decomposition
3. Mesh Scheme Selection
4. Mesh Size/Interval Assignment
5. Mesh Generation
6. Mesh Quality Analysis

Sandia National Laboratories
Boundary Conditions

• Three primary types of boundary condition groupings:
  – *Element Block*: not really a BC; groups elements for purposes of material type identification
  – *Nodeset*: for assigning loads on nodes in the model, e.g. Temperature, Force
  – *Sideset*: for assigning loads on faces in the model, e.g. Pressure
• Import into Ansys as named *components*, then use to assign loads to model there

• Examples:
  
  Block 100 volume 1
  Nodeset 2001 surface 1 2 3 curve 100 volume 3
  Export genesis ‘test.g’
Transferring Mesh to Ansys

- Write mesh to “Exodus” or “Genesis” file ‘basename.g’
- Translate to Ansys format: ‘exoans basename.g bname’

<table>
<thead>
<tr>
<th>Concept</th>
<th>Exodus</th>
<th>Ansys</th>
<th>File</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>Nodes</td>
<td>Nodes</td>
<td>bname.node</td>
</tr>
<tr>
<td>Elements</td>
<td>Elements</td>
<td>Elements</td>
<td>bname.elem</td>
</tr>
<tr>
<td>Material type</td>
<td>Elem Block xxx</td>
<td>Component EBxxx</td>
<td>bname.e.set</td>
</tr>
<tr>
<td>Nodal BC</td>
<td>Nodeset xxx</td>
<td>Component NSxxx</td>
<td>bname.n.set</td>
</tr>
<tr>
<td>Area BC</td>
<td>Sideset xxx</td>
<td>Component SSxxx</td>
<td>bname.s.set</td>
</tr>
</tbody>
</table>

- Read into Ansys:
  - Enter pre-processing mode & select default material
  - “File->Read Input” from file ‘bname’ (includes other files by reference)

- Select->Comp/Assembly->Select Comp/Assembly
  - Proceed as before, with BC’s, IC’s
The CUBIT/CLARO Mesh Toolkit
CLARO GUI

Graphics controls (wire, shaded, etc.)
Mouse controls (rotate, pan, etc.)
Picking controls
Primary mode (geom, mesh, props, export)
Secondary mode (depends on primary mode)
Other mode-dependent sub-windows

Graphics window
Command window
Example

- Start claro  > ~tautges/cubit/claro
- Create cylinder  > Cyl rad 1 height 1
- Click & drag mouse in graphics window  > [Left MB]: rotate
  > [Middle MB]: pan
  > [Right MB]: zoom
- Select tet mesh scheme  > Volume 1 scheme tetmesh
- Set mesh size  > Volume 1 size 0.5
- Mesh  > Mesh volume 1
- Check mesh quality  > Quality volume 1
- Export  > Export genesis ‘mymesh.g’
Some More Useful CUBIT Commands

> List surface 3

• Pick geometry in graphics window
  – <Ctrl>-Left on geometry in graphics window
• Switch picking to entities of dimension 0/1/2/3
  – With mouse in graphics window, type 0/1/2/3

> Delete mesh

> Curve 3 size .2

• Set node-based BC group on 2 surfaces
  > Nodeset 100 surface 2 10

• Import solid model from file
  > Import acis
  ‘/pong/usr1/t/tautges/cubit/spindle.sat’
To Learn More…

- Plenty of small project topics for your final project
- If you’re interested in programming, also some hourly work I have
- Meshing Research Corner:  
  http://www.andrew.cmu.edu/user/sowen/mesh.html
- Mesh Generation & Grid Generation on the Web:  
  http://www-users.informatik.rwth-aachen.de/~roberts/meshgeneration.html

http://www.gotmesh.org
Next Time…

• Will learn more about how various meshing algorithms work
• Will mesh a simple example & transfer mesh to Ansys
• Next assignment:
  – Generate simple mesh w/Claro & analyze in Ansys