Roopesh Mathur
GAMBIT Tips and Tricks
May 24, 2006

2006 CFD Summit | Share . Inspire . Learn . Connect

Agenda

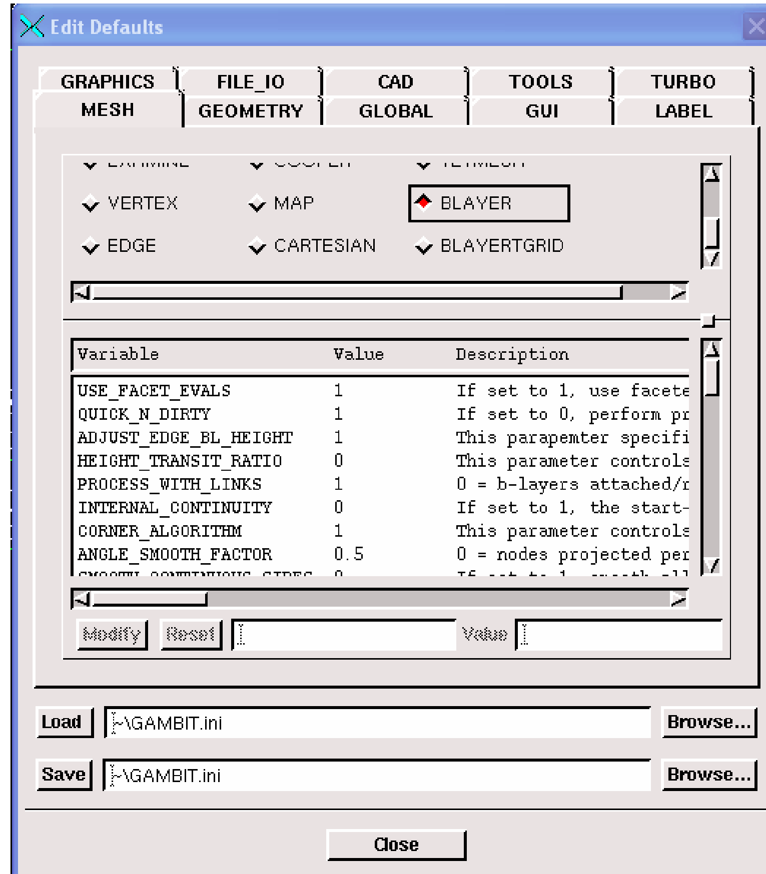
- GAMBIT Overview
- CAD Import
- Geometry Operations
- Meshing Operations
- Meshing Strategies
- Summary and Conclusions

GAMBIT Overview

- GAMBIT Defaults
- Journal Files
- Forms and Picking Geometry

GAMBIT Defaults

- GAMBIT Defaults are environmental variables which affect:
  - User Environment
  - Meshing Characteristics
  - Geometry
- Use **Edit Defaults** to customize default settings
  - For example, set GLOBAL.GENERAL.SOLVER = 2 for Polyflow use
  - Save defaults in **`~/Gambit.ini`** file



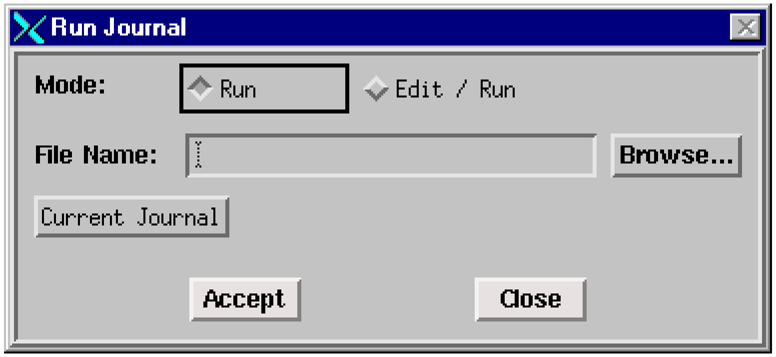
- More information on the defaults is in the Gambit Defaults Guide in GAMBIT Documentation.

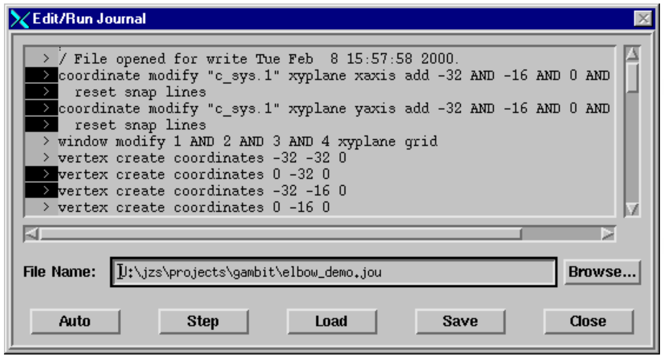
FLUENT
2006 CFD Summit

Journal Files

- Journal File:
  - Executable list of Gambit commands
    - Created automatically by Gambit from GUI and TUI.
    - Can be edited or created externally with text editor.
    - Executed in Run Journal or Edit/Run (Interactive mode)
    - Small text files – easy to store or e-mail.

  Uses:
  - Can be parameterized, comments can be added
  - Easy recovery from a crash or power loss
  - edit existing commands to create new ones





FLUENT
2006 CFD Summit

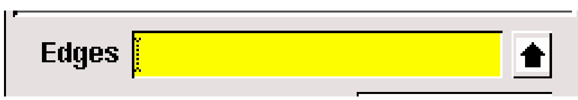
Journal Files

- Extracting lost journal file from existing database file
  `gambit –res(torejournal) journalfile –id database`

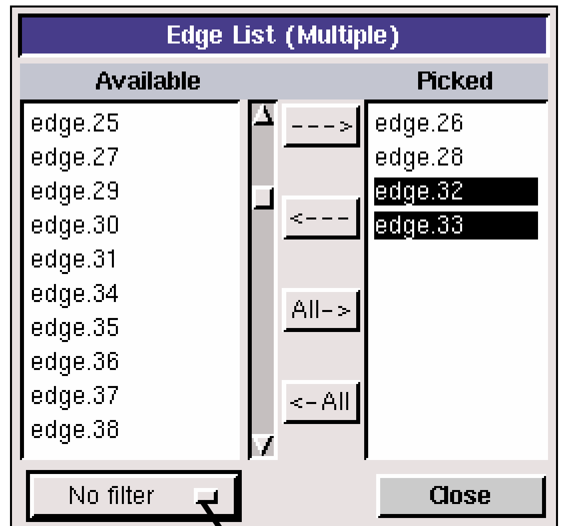
- Tips to minimize impact when running old journal files
  - Use default GUI.GENERAL.JOURNAL_ENTITY = 1 or 2 to generate ID independent labels, if possible
    - 0 – journal original labels (default)
    - 1 – journal lastid, i.e. creation order
    - 2 – journal entity location
  - Set GEOMETRY.VOLUME.BOOLEAN_METHOD = 0 to use old Boolean/imprinting method
  - Set GEOMETRY.GENERAL.REAL_LABEL_CHANGE_MINIMIZATION = 0 when running Gambit 2.0 journal files
  - Set GEOMETRY.FACE.VIRTUAL_FACE_SPLIT_METHOD = 0 to get old behavior

Picking Geometric Entities: Pick Lists

- Pick Lists
  - Open the Pick List by clicking on the arrow and move entities between Available and Picked lists.

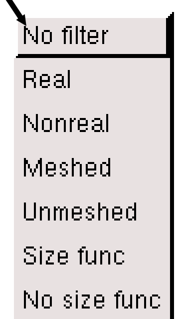


Available Entities



Picked Entities

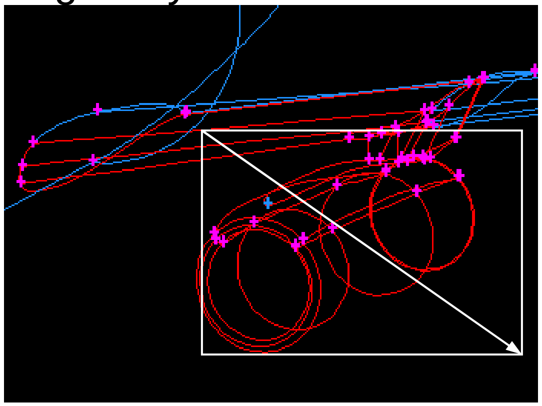
- Using picking filter
  - Picking based on real/virtual meshed/unmeshed entities



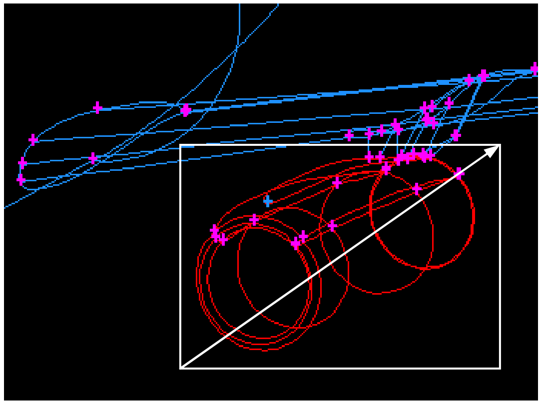
Picking Geometric Entities: Mouse

- Picking by dragging a box over entities

Picking 'partially included' entities when dragging box diagonally down



Picking 'completely included' entities when dragging box diagonally up



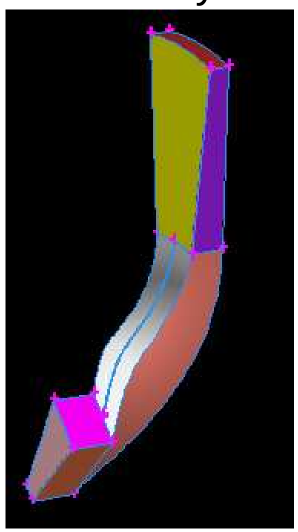
- Using mouse and cursor: *Shift* + …
  - Left mouse: initial pick, additional picks
  - Middle mouse: modify pick
  - Right mouse: 'Apply' or next list box

FLUENT
2006 CFD Summit

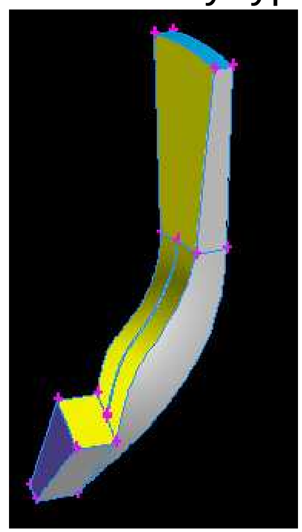
Color by Zone Type/ID

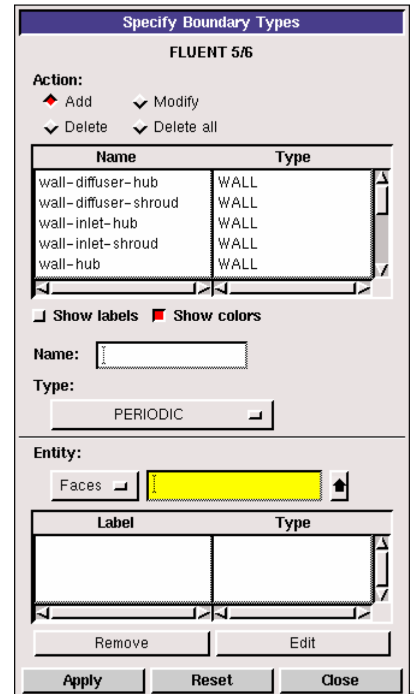
- Option to color zones by zone type or by zone ID
  - Similar to Fluent and TGrid
  - GRAPHICS.GENERAL.ZONECOLOR_TYPE = [0, 1]

0 – color by ID



1 – color by type





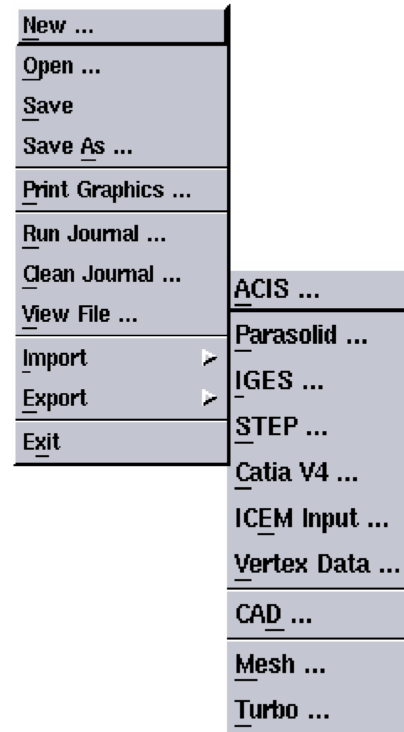
- Change the default black background color
  - GRAPHICS.GENERAL.WINDOWS_BACKGROUND_COLOR

CAD Import

- CAD Import Options in GAMBIT
- CAD Interoperability Issues

CAD Import

- GAMBIT offers many options for importing geometry from CAD systems.
  - Direct Translation options for standard CAD kernels (ACIS, Parasolids, CATIA)
  - Standard translation options (STEP and IGES)
  - ProE options: Direct and STEP import
  - Faceted/Mesh Import options: FLUENT, STL etc.



- GAMBIT has specialized tools for dealing with CAD Geometry issues.
  - Translation problems, defect removal and simplification issues

FLUENT
2006 CFD Summit

CAD Interoperability Issues

- CAD interoperability, or the ability to share a CAD model across different applications has been a challenge for industrial engineers.

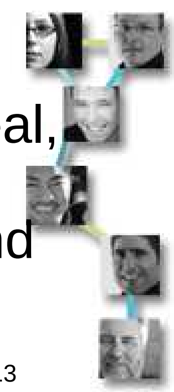
- The key issues affecting CAD interoperability are:
  - **Model quality in the originating CAD system**: missing parts, invalid definition, lack of connectivity, integrity.
  - **System-specific Semantics**: Customized features and localized flavors translate poorly across systems.
  - **Differences in tolerance:** CAD systems often use a loose (1e-03) tolerance, which is not sufficient for GAMBIT (1e-06).
  - **Limitations of translation:** Data types do not have one-to-one correspondence across different systems with different definitions, which leads to approximation by the translators and receiving system.

- This can lead to “dirty” or disconnected geometry when imported into GAMBIT.

CAD Interoperability Strategies

- Some specific strategies users can pursue to reduce interoperability issues:
  - Upstream end (originating CAD system)
    - Use of a tighter or absolute tolerance (closer to 1e-06)
    - Use of solid or native CAD models rather than wireframe.
    - If using neutral formats like STEP, importing the data back for a loop test to ascertain that the solid model can be recreated.

    - Methods which are specifically developed for translation:
    - STEP Export/Import
    - Feature based translation
    - Direct Translation/Interfaces with upstream CAD kernel (ProE/ others under development)
  - Downstream end (GAMBIT): Tolerant Modeling, Smooth/Heal, Cleanup Tools
  - Please refer to the "CAD for CFD" document on the USC and the CAD Connections lecture.



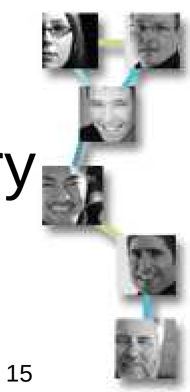
Geometry Operations

- Checking geometry
- Repairing Geometry
  - Smoothing
  - Healing
- Geometry Construction tools
- Booleans
- Cleanup Tools: Virtual Operations

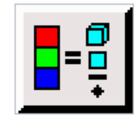
Checking Geometry

- Geometry imported or created needs to be checked for integrity, connectivity and completeness.
  - Integrity: corrupt or degenerate volumes, faces or edges
  - Connectivity: Disconnected faces or edges, slivers.
  - Completeness: Missing volumes, faces or edges

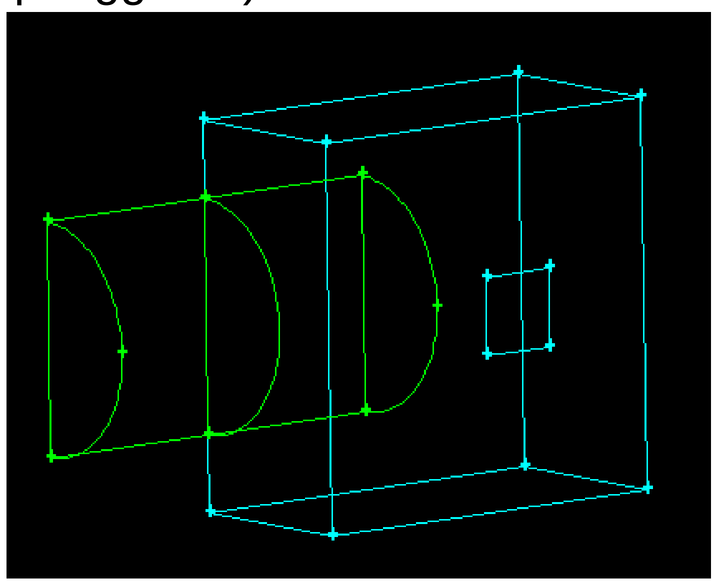
- Geometry can be checked using visual or ACIS-based checks.
  - Visual Checking: Color by Entity or Color by Connectivity
  - ACIS based Checks: Checks selected geometry for topological and geometric consistency.



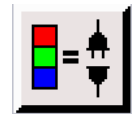
Color by Entity



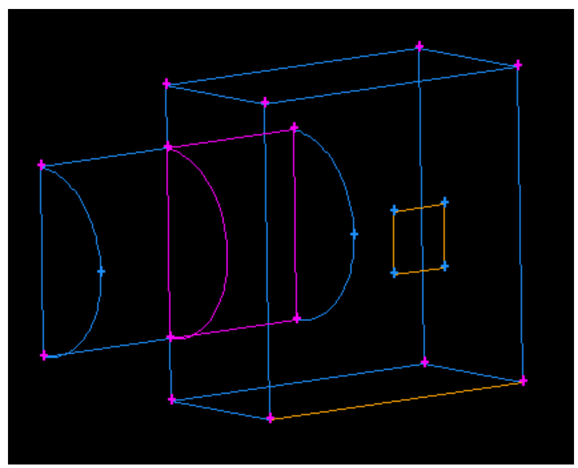
- Check import of volumes/faces using entity based coloring
  - Vertices (white), edges (yellow), faces (light blue), volumes (green), groups (dark green)
  - Virtual/faceted geometry identified by label (v_*/f_*), picking filter, and color coding
    - Vertices (darkkhaki), edges (peru), faces (dodgerblue), volumes (springgreen)



Color by Connectivity

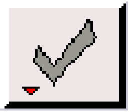


- Check connectivity using connectivity based coloring
  - White: Stand-alone vertex or edge
  - Orange: Vertex connected to only one edge, or edge connected to only one face (i.e. unconnected face!)
  - Blue: Vertex connected to exactly two edges, or edge connected to exactly two faces (i.e. connected face)
  - Magenta: Vertex connected to three or more edges, or edge connected to three or more faces (e.g. internal face)

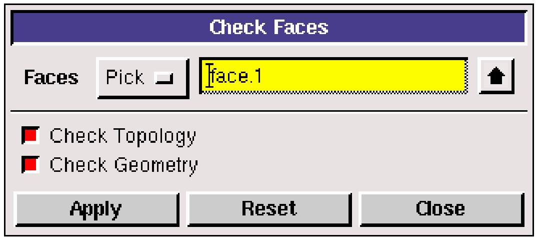


ACIS Based Checks

- Checks for geometrical and topological inconsistencies



  - Geometric consistency: Assesses model parametric/geometric continuity and distances between connected geometry.
  - Topology consistency: Assesses model for consistency in organization (how geometry is constructed or virtual guest-host relationships)
  - Setting GEOMETRY.GENERAL.REAL_GEOMETRY_CHECK_SUMMARY to 1 will produce a detailed report.



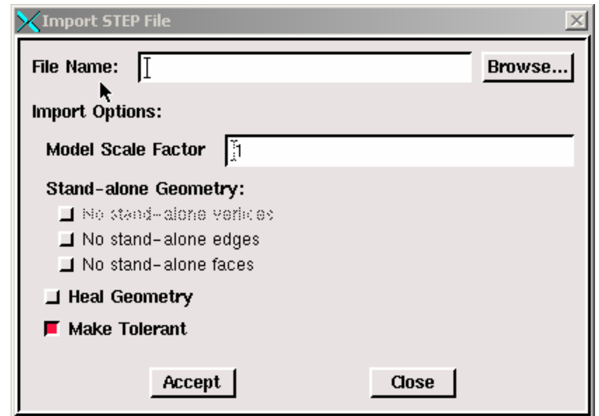
  - Geometry not C1 or G1
    - Indicates lack of parametric/geometric continuity
    - Real operations (booleans, splits, sweeps) may fail
    - Can use virtual operations as fallback
    - Geometry errors do generally not prevent meshing. **If the model can be shaded, then it can likely be meshed.**

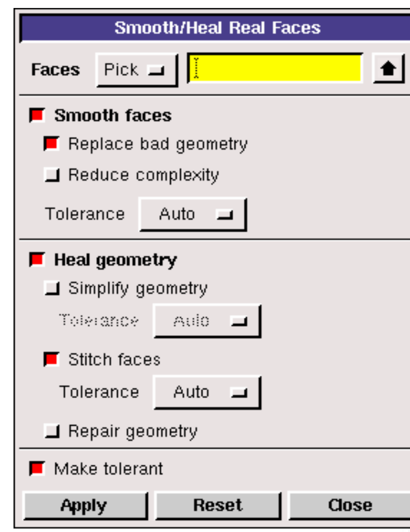
Fixing Geometry Problems

- Problems uncovered by the visual and ACIS checks can be fixed:
  - Fix real geometry problems using tolerant modeling, smoothing or healing the unconnected entities, or by regenerating entities.
  - Fix topology problems by deleting and reconstructing entities.
  - Fix virtual/faceted geometry problems by deleting (hiding) and reconstructing entities.

Tolerant Modeling

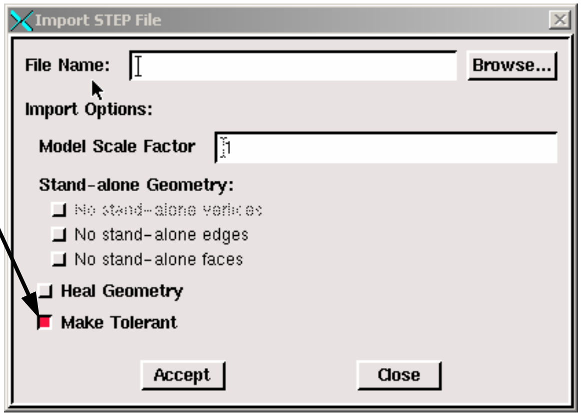
- Every CAD system uses a minimum numerical precision (tolerance) for describing the geometry.
  - Coordinates of vertices/points, length of edges, size of faces, volumes.
- The ACIS kernel in GAMBIT uses a high precision tolerance of 10^{-6} (6 decimal places), which may not be matched by the CAD software in which the geometry was created.
  - Leads to disconnected edges and faces.
- Tolerant Modeling is a technique which uses a variable or floating tolerance to connect geometry.

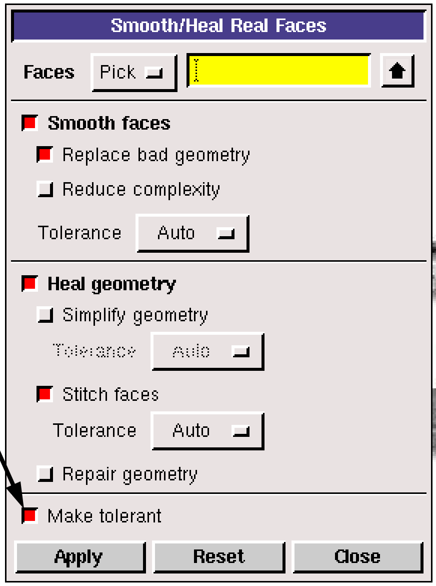




Tolerant Modeling

- Available by activating the Make Tolerant option during CAD model import.
  - Also available after import inside the Heal Face (or Volume) Form.
- Application
  - All Geometry files
  - Relatively large gaps
  - Real ACIS volumes generated during import.
- Boolean operations subsequently possible
  - Adding/Subtracting additional geometry
  - Volume Extraction
  - Retaining only ½ or ¼ of model
  - Volume decomposition for better meshing

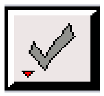




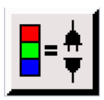
FLUENT
2006 CFD Summit

Smooth/Heal Real Geometry

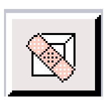
- Geometry imported from other CAD systems can lack the required accuracy and precision to render valid or connected ACIS geometry.
  - This results from numerical limitations in original CAD system, neutral file formats, or differences in tolerances between CAD systems and ACIS.



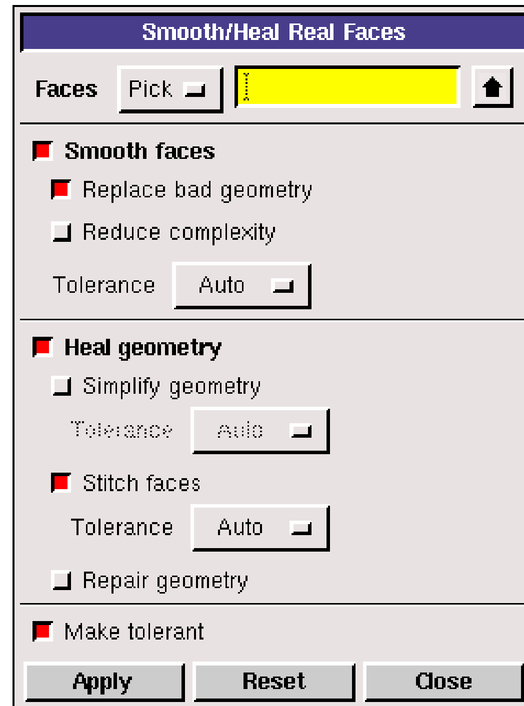
  - Use the check command to verify integrity of geometry/topology.



  - Check the connectivity of geometry using connectivity-based color coding.



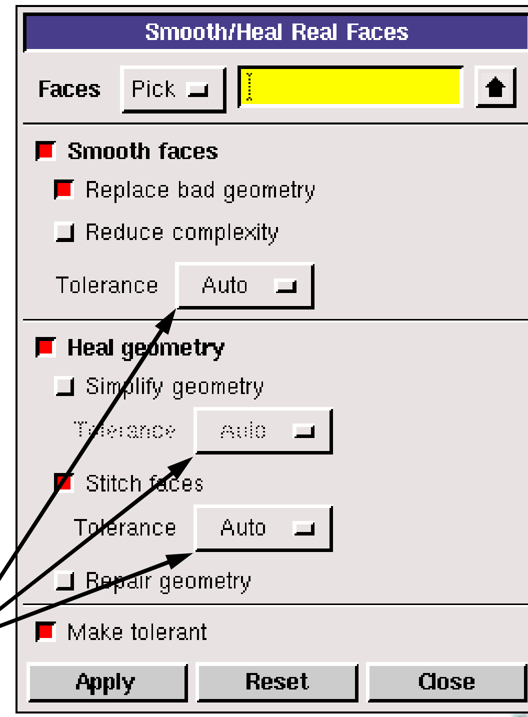
- Smoothing and Healing of Real geometry can help.
  - Smoothing is used to remove discontinuities in geometry.
  - Healing is used to detect and correct inaccuracies in model geometry due to different tolerances and translator limitations.
  - Available in the Smooth/Heal forms or as an option during CAD Import.



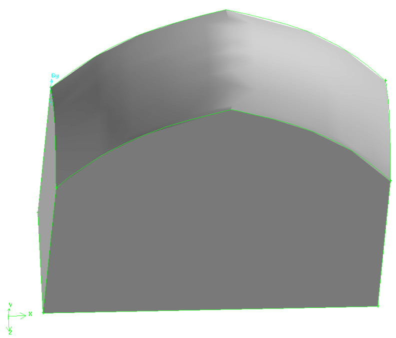
FLUENT
2006 CFD Summit

Smoothing and Healing Options

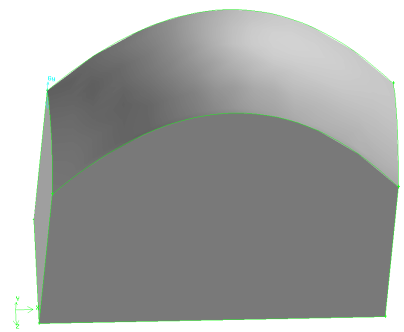
- Smooth Options:
  - Replace Bad Geometry: Removes discontinuities or “kinks” in geometry.
  - Reduce Complexity: Simplifies the spline representation of geometry by reducing the number of control points.
- Healing options:
  - Simplify Geometry: Converts spline geometry to analytic geometry, whenever possible.
  - Stitch Faces: Stitch together faces being healed to form one or more volumes.
  - Repair Geometry: Changes the definition of the edges and faces of geometry so that the model “fits together” well, especially at boundaries.
- A tolerance is specified for some operations
  - The maximum distance between old and new geometry.
  - The maximum size of the gap between faces for Stitch faces.
  - Auto: Automatically calculates the tolerance based on the size of the geometry being smoothed/healed.
  - Manual: User can set tolerance (10^{-3} by default).



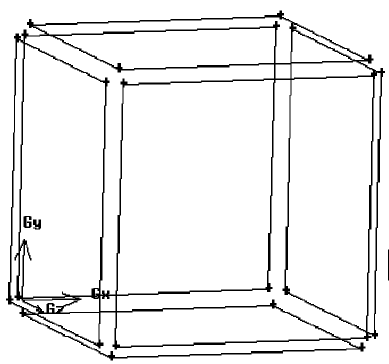
Smoothing and Healing Examples

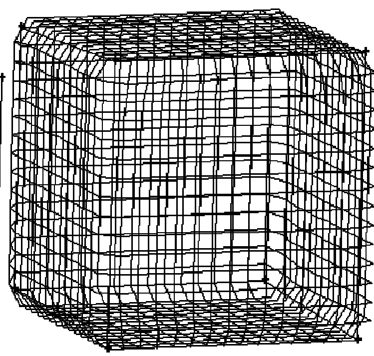


Face smoothing

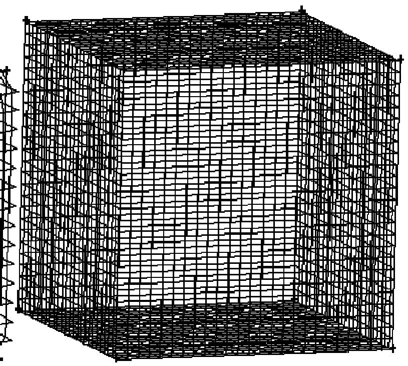


Healing Faces





Stitch faces options



Stitch faces, Repair geometry options

Real Geometry Operations

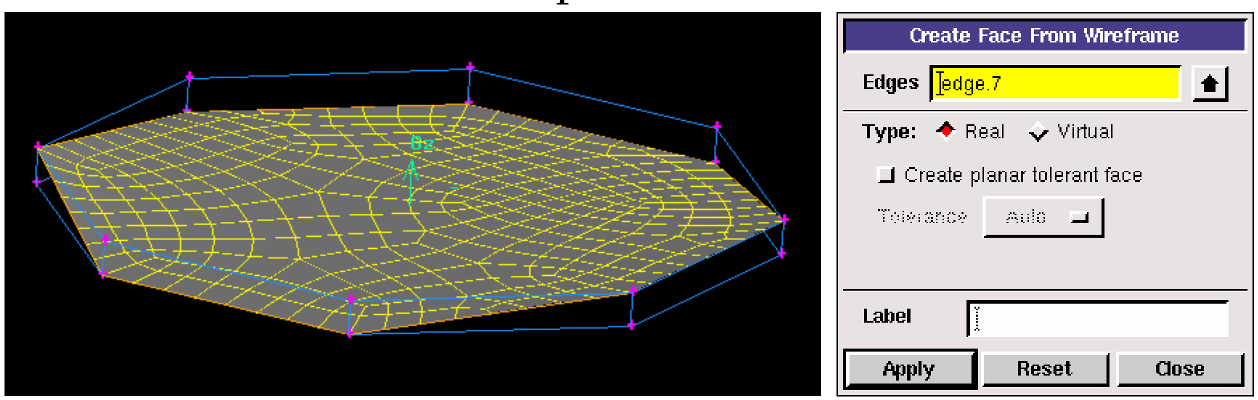
- Real geometry operations are those which operate on Real Geometry and result in the creation of real geometry, i.e., those described by precise mathematical formulas.

- These operations include:
  - Real face from wireframe
  - Volume stitch
  - Booleans (subtract, unite, intersect) and split

Real Face From Wireframe

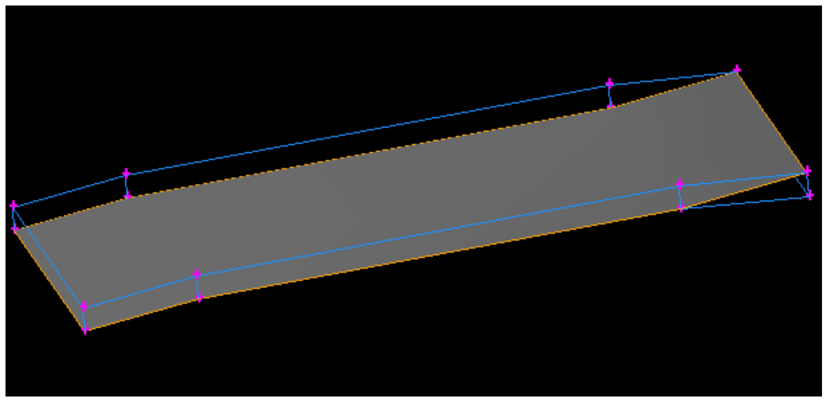
- Create face from wireframe
  - Coplanar edges
    - Any number of coplanar edges within ACIS tolerance can be stitched into a real face
  - Non-coplanar edges
    - Any 3 or 4 sided non-planar loop can be stitched into real face
    - Loops with 5 or more non-planar edges can be stitched into real face if
      - Edge loop is convex
      - Edges don't turn 'excessively'

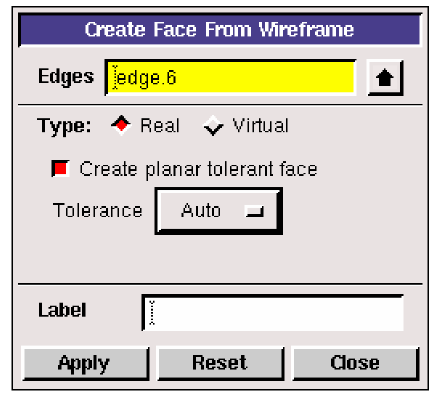
Convex Non-coplanar Face Creation



Create Planar Tolerant Face Option

- A planar tolerant **real face** creation option was introduced in Gambit 2.2 for non-coplanar edge loops, where the edges are close to being co-planar.
  - Tolerant option ignored if ACIS is able to create non-tolerant face.





Maximum tolerance reported in transcript window

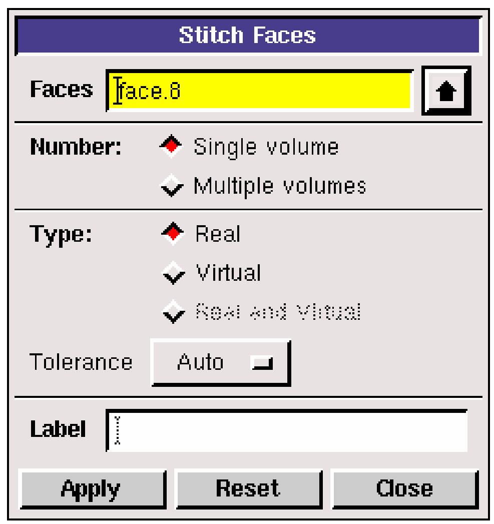
```
Command> face create wireframe "edge.1" "edge.2" "edge.25" "edge.21" "edge.26" \
  "edge.6" "edge.7" "edge.8" real tolerance -1
WARN:  The given set of edges deviate highly from the constructed planar face.
It is possible that subsequent operations with this face may fail.
The ACIS (tolerant) face has tolerance value of 0.047599.
Created face: face.11
```

- Tolerance should be smaller than intended mesh size
- Turn off final projection of mesh from facets to geometry if tolerance is larger than mesh size (MESH.FACE.PROJECT_TO_SURFACE = 0)

FLUENT
2006 CFD Summit

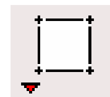
Real Volume Stitch

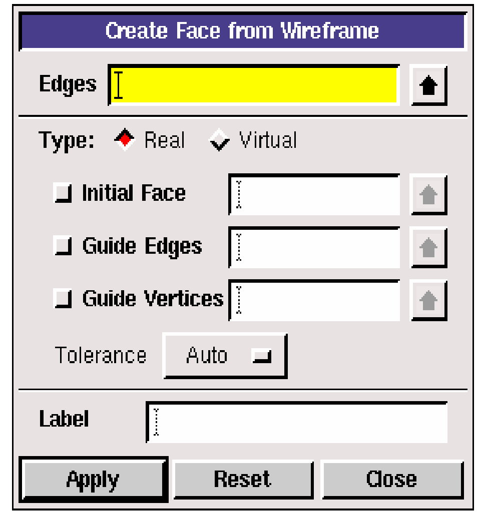
- Volume Stitch is a tolerant operation which allows you to stitch faces into volumes, with options to:
  - Stitch single or multiple volumes
  - Create real, virtual or real and virtual volumes in a single step.
  - Automatic/Manual Tolerance Control.



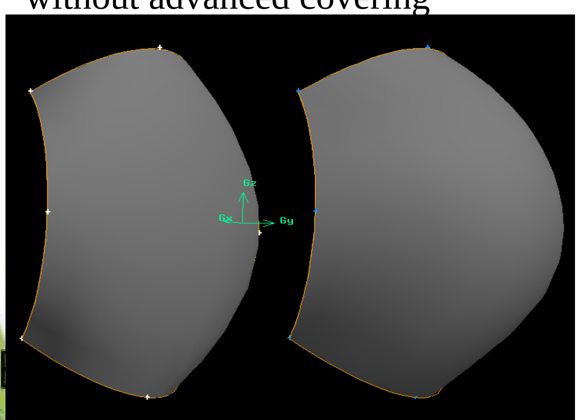
Advanced Covering

- Advanced Covering is a new capability to construct new geometry based on existing geometry or mesh.
    - Better quality faces using existing faces, edges or vertices as a guide.
    - Virtual to real conversion using existing tri surface mesh on any arbitrarily shaped single loop face.

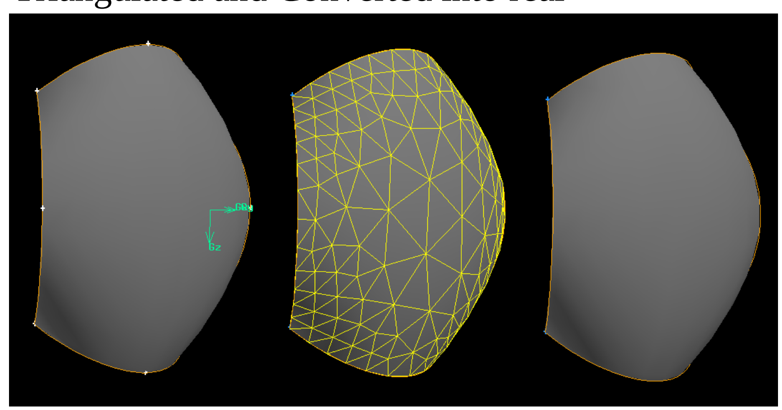




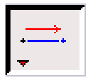
Example of face creation with and without advanced covering



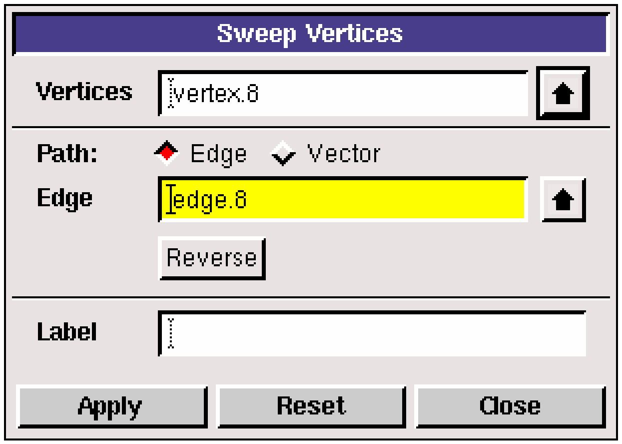
Example of Original virtual face, Triangulated and Converted into real

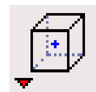


New Geometry Operators

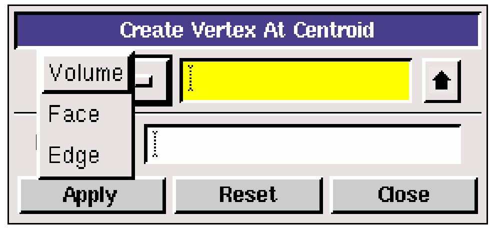


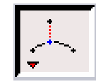
Sweep Vertices



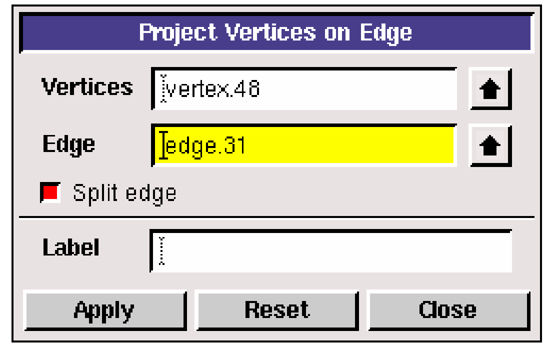


Create Vertex at Centroid

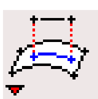




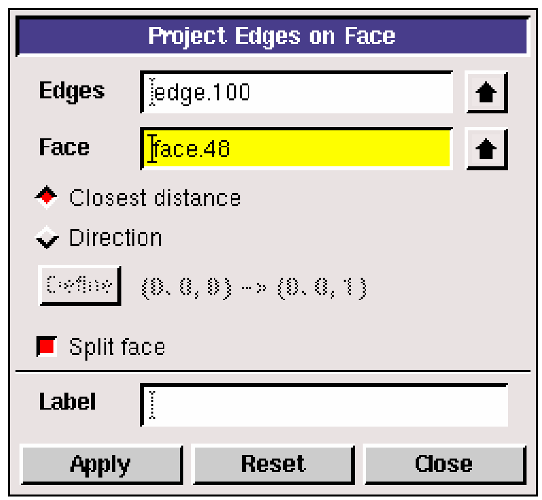
Real Vertex-to-Edge Projection
Split Edge option (Real)



Real Edge-to-Face Projection



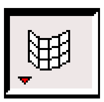
- Using “Closest distance”
- Multiple Edge pickers
- Split Face option (Real)

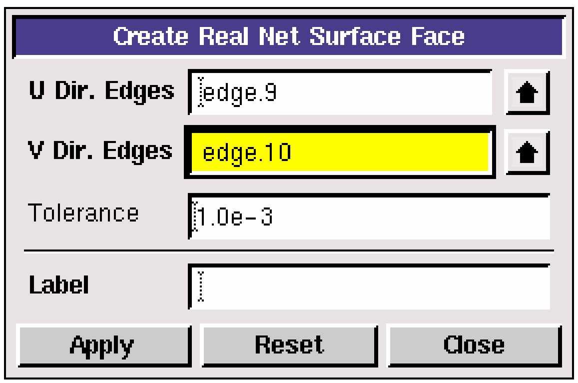


FLUENT
2006 CFD Summit

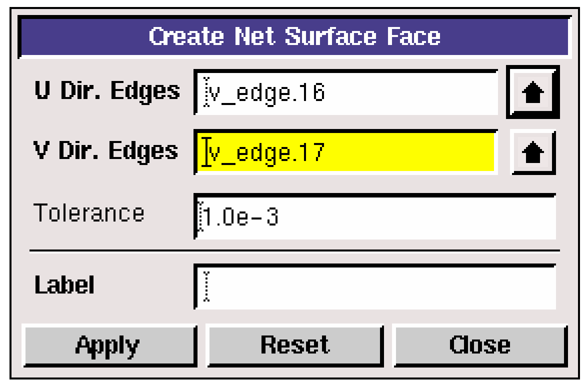
New Virtual Geometry Capabilities

- Virtual geometry can now be used in all edge and face construction tools.
  - Example: Net surface creation can be done using existing virtual edges.



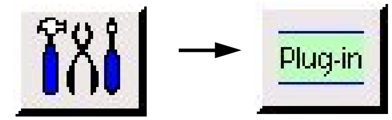


GAMBIT 2.2

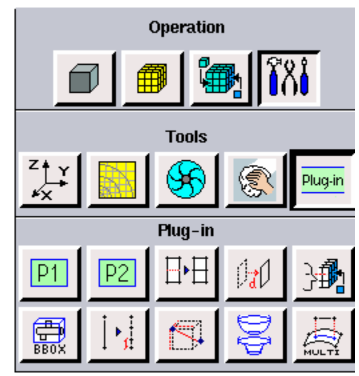


GAMBIT 2.3

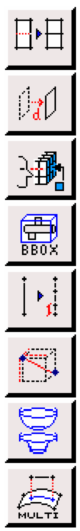
Plug-In Tools



- Plug-ins are extra tools which can be added to GAMBIT.
  - Download plugins to:
    - **`\FLUENT.INC\Gambit2.3.x\plugins`** (Windows)
    - Home directory (UNIX/LINUX)



  - Currently developed plugins



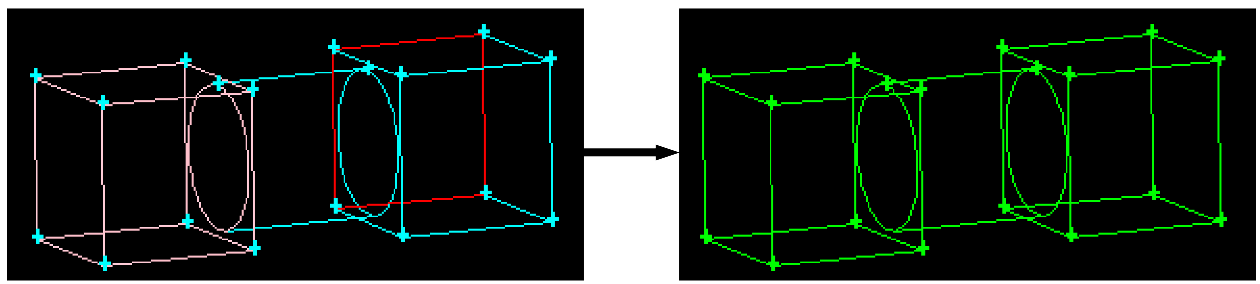
    - Split multiple faces with a face
    - Create a face via offset
    - Control visibility by zone
    - Create a brick based on the bounding box for the current geometry
    - Multiple splitting of edges based on equal spacing or actual length
    - Calculate distance between two vertices
    - Convex or concave pipe size transitions
    - Project multiple edges onto multiple faces

Option to Stitch Single or Multiple Volumes

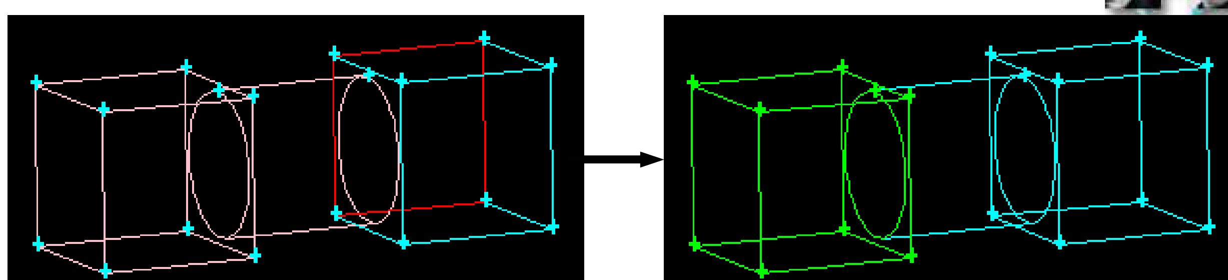
- Option to stitch single or multiple volumes
  - Automatic addition of missing (connected) faces for single volume
  - Extra faces which do not form a closed set automatically discarded for multiple volumes
  - Single volume stitch **can handle voids** and dangling faces.

Single volume stitch

(adds faces automatically)



Multiple volumes stitch (discards extra faces)



Real Boolean Operations

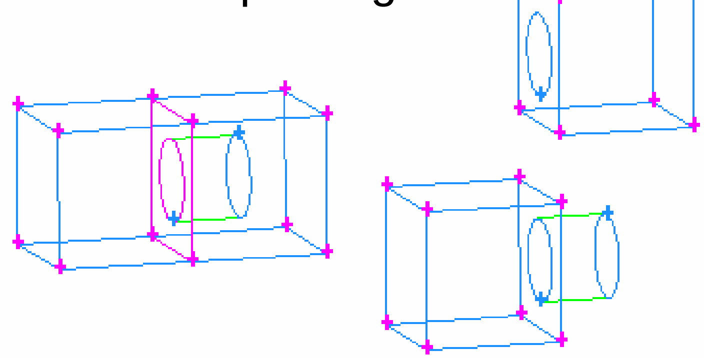
- Why Boolean operations can fail:
  - Poor geometric quality, complex topology
  - Difficulties in finding imprints of connected entities
    - Two imprinting algorithms available

      GEOMETRY.VOLUME.BOOLEAN_METHOD = 0, 1
      - A value of 0 turns on global imprinting (more difficult)
      - A value of 1 (the default value) allows local imprinting only.
        - Generally stable but requires subsequent bidirectional split.
  - NURBS surfaces do not represent cylinders and spheres exactly. Boolean between ‘coincident’ analytic and NURBS surfaces are unpredictable

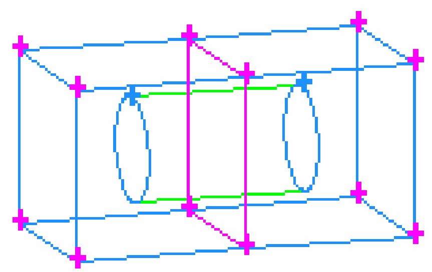
Imprinting in Boolean Operations

GEOMETRY.VOLUME.BOOLEAN_METHOD = 1

Local imprinting

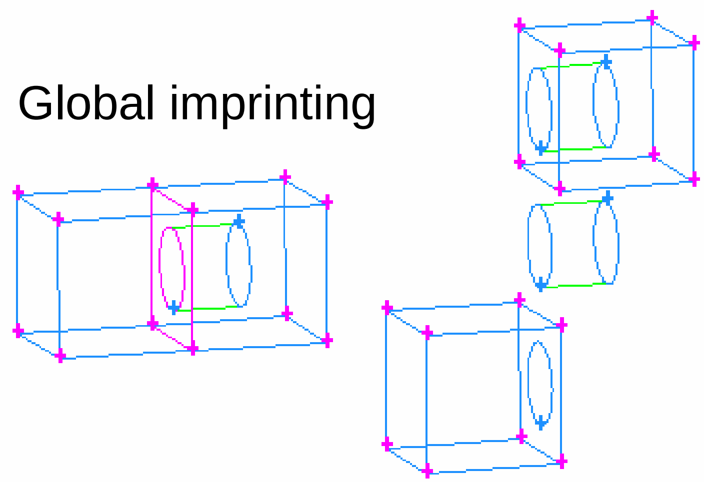


Unite cylinder with left cube



GEOMETRY.VOLUME.BOOLEAN_METHOD = 0

Global imprinting



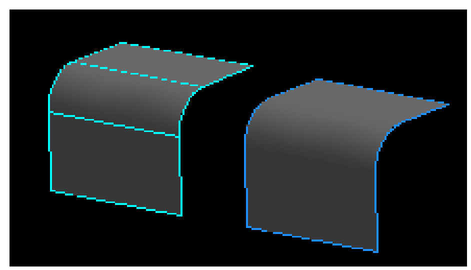
FLUENT
2006 CFD Summit

Failing Real Split Operations

- What to do if real split operation of entity A with B fails
  - Copy/Heal option
    - Copy B (and heal if needed), delete the original B
    - Perform the split operation, reconnect before meshing
  - Intersection/Subtraction option
    - Intersect A with B while retaining A
    - Subtract the result (intersection) from A while retaining the tool
    - Connect before meshing
  - Clean geometry through mesh export/import

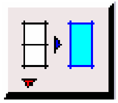
Virtual Geometry Operations

- ‘Overlay’ technology
  - Tool of choice to **clean-up and simplify** imported geometry
  - Needed when tolerant modeling and healing/smoothing are not sufficient
  - Modifies size and shape of boundary surfaces while being constrained by underlying ‘host’ geometry definition
    - Represents same geometry but different topology
    - Can be derived from real, virtual, or faceted geometry
  - Limited use for Boolean operations
  - Semi-automatic virtual clean-up tool available

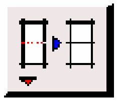


Three real faces merged into one virtual face

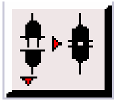
Virtual Geometry Operations



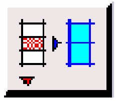
- Merge: Replaces two connected entities (edge/face/volume) with a single virtual entity
  - Only volumes can be merged in real



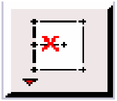
- Split: Partitions an individual entity (edge/face/volume) into two or more connected virtual entities



- Connect: Combines two individual unconnected entities (vertex/edge/face) into a single virtual entity
  - Connect entities by connecting lower order topology
  - Virtual connect can bridge relatively large gaps in model, larger than ACIS tolerance of 1.0e-6



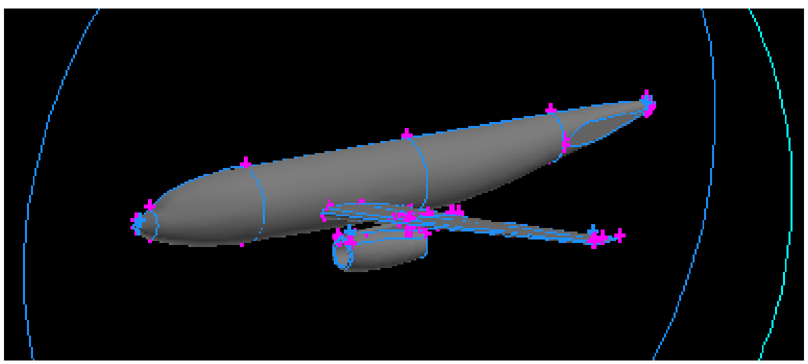
- Collapse: Splits a face and merges the resulting with two neighboring faces

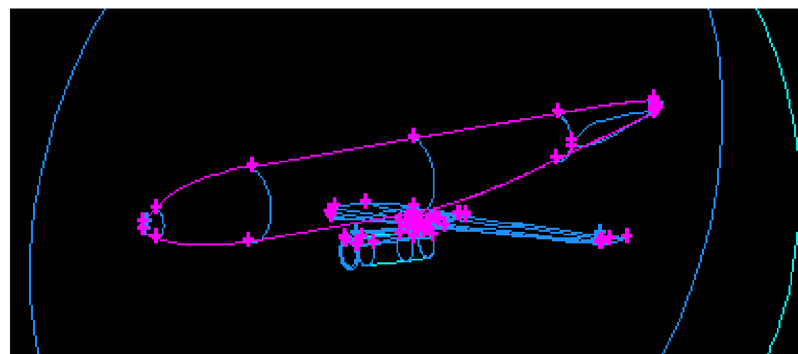


- Simplify Faces: Removes dangling edges

Face Connect with T-junction splits

- New functionality to ensure connectivity between non-aligned faces.
  - Utilizes projections, splits, and connects
  - For real and virtual geometry, resulting geometry is always virtual
  - Easy connecting of poorly matching faces.
  - Allows imprinting of complex half geometry on symmetry planes.

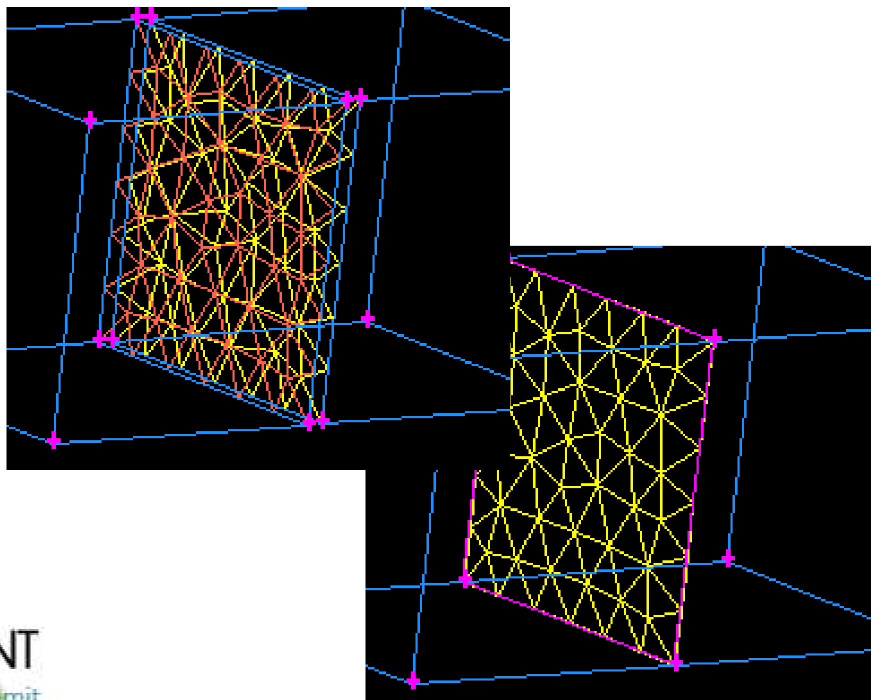


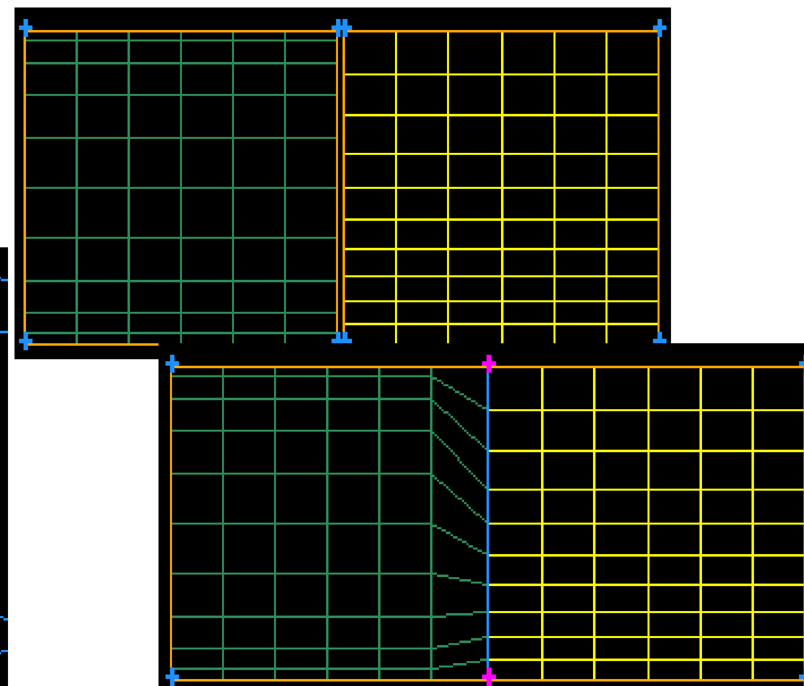


FLUENT
2006 CFD Summit

Connect Preserving Mesh

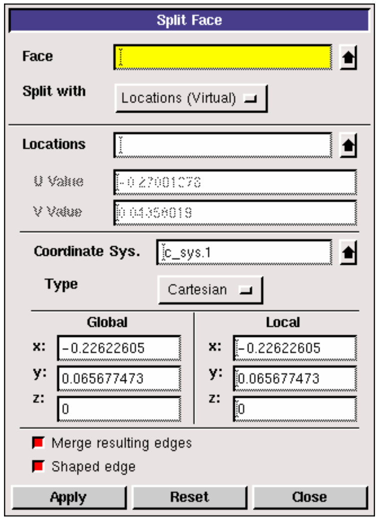
- Preserve higher topology mesh during edge and face connect
  - Mesh on unconnected entities must be topologically equivalent, but **can have different grading**
  - Might require mesh smoothing after connect operation
- Allows easier meshing of geometry of repeated patterns
  - Copy geometry using ‘mesh unlinked’ option before connecting.





Virtual Face Splits

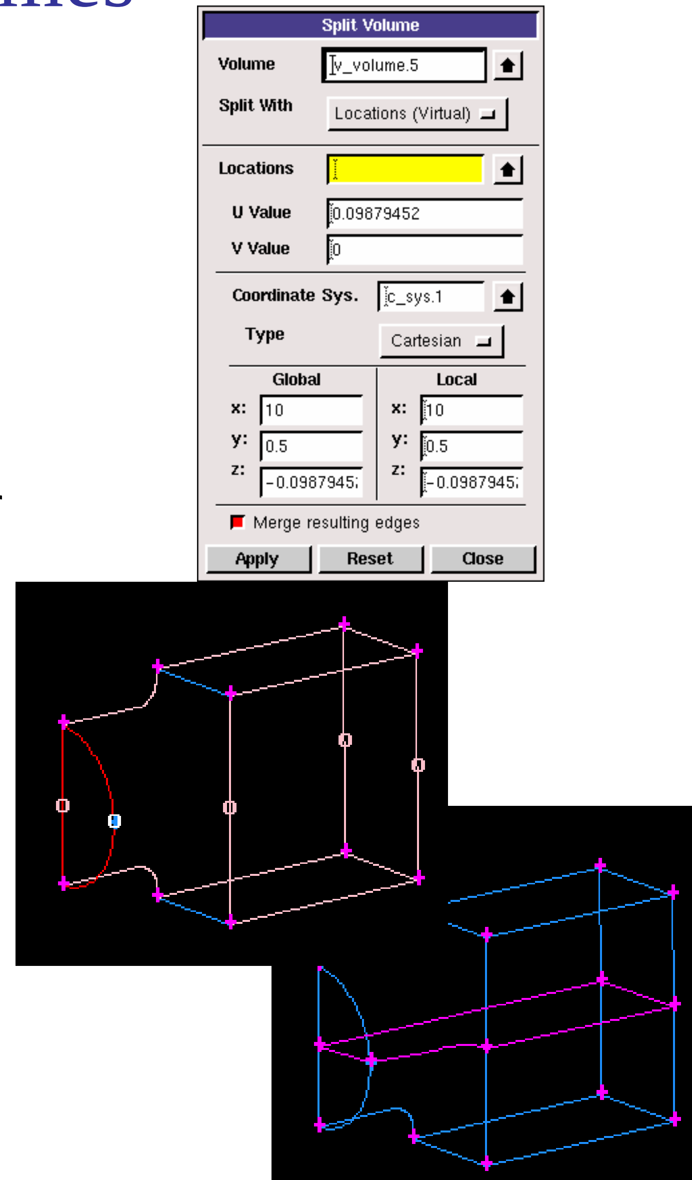
- Face splits by
  - Face (Virtual)
    - Set GEOMETRY.FACE.VIRTUAL_FACE_FACE_SPLIT to 1 in defaults
    - Bi-directional option
    - Try exporting surface meshes and splitting imported faceted faces if virtual face split fails
  - Edges (Virtual)
    - Endpoints of splitting edge need to be connected to edges of face
    - Tolerance for edge proximity to face
  - Vertices (Virtual)
    - Tolerance option for vertex proximity to face
    - Can produce dangling edges
    - ‘Shaped edge’ option
  - Locations (Virtual)
    - Locations snap to edges near boundaries
    - Manually drag and position temporary ‘Locations’
    - ‘Shaped edge’ option



FLUENT
2006 CFD Summit

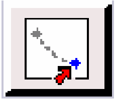
Decomposing Virtual Volumes

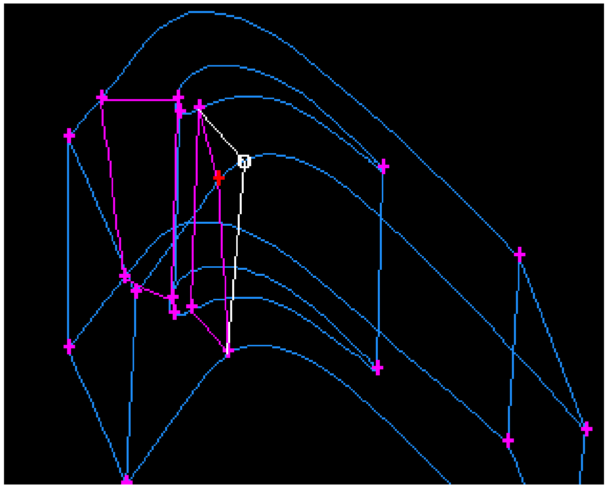
- Volume splits by
  - Faces (Virtual)
    - All split tool faces must be connected together
    - Must share boundary edges with volume boundary
    - If volume split fails (rarely), make volume invisible (keep lower topology visible) and stitch together desired volumes
  - Locations (Virtual)
    - Easy tool to split virtual volumes
    - Manually drag and position temporary 'Locations'
    - Splitting volumes with voids not supported

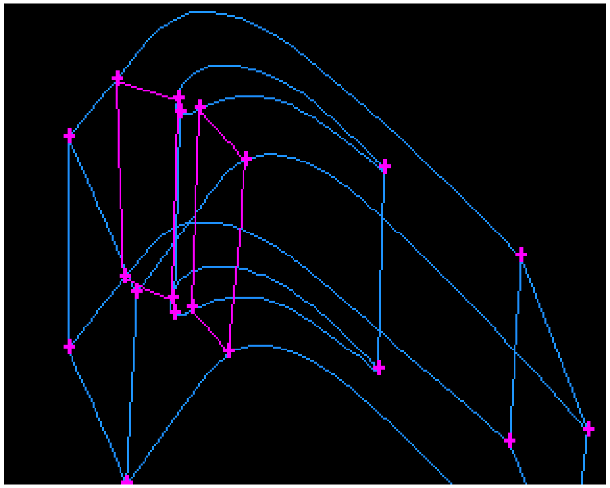


Adjusting Virtual Geometry

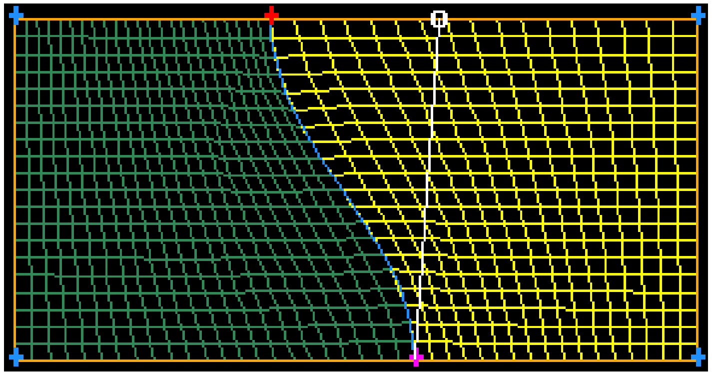
- Adjust virtual geometry using the ‘slide virtual vertex’ tool

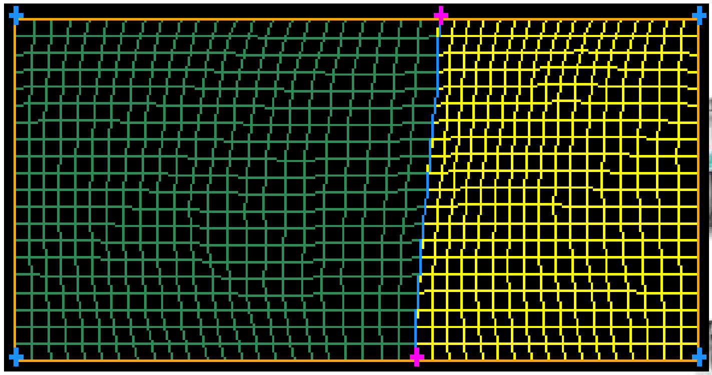






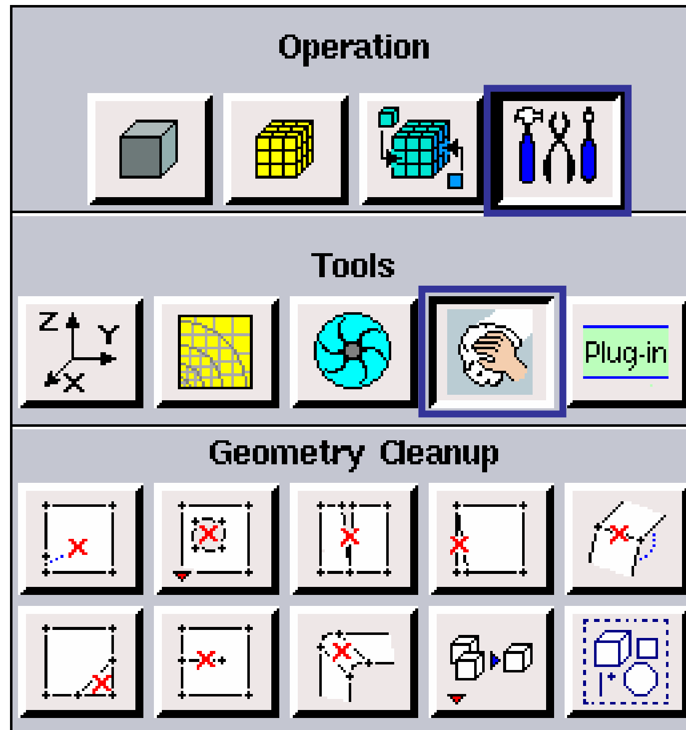
Vertices can be adjusted after meshing (might require subsequent smoothing of mesh)



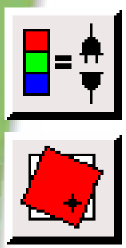


Semi-Automated Cleanup Tools

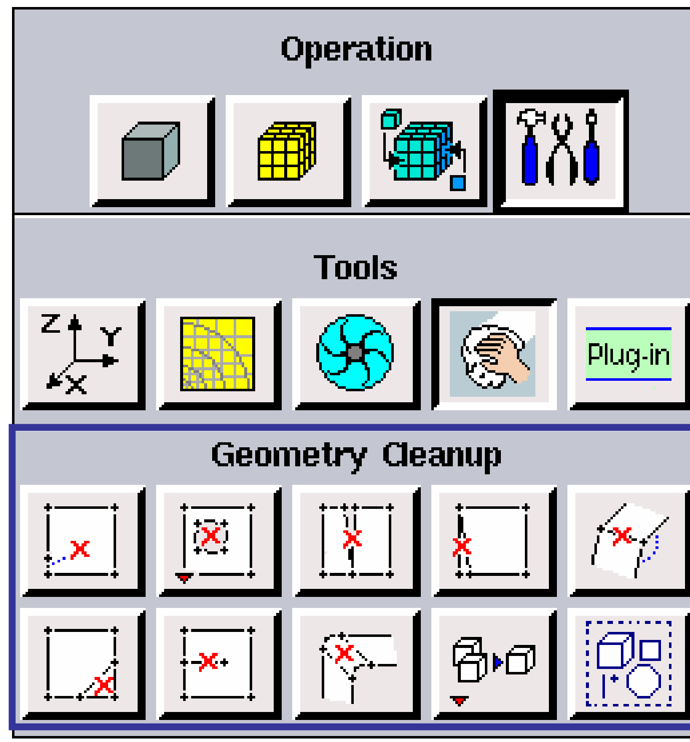
- Clean up on models containing a large number of faces can be tedious.
- Cleanup Tools can semi-automate this process using virtual and real operations.
  - Locate problem areas
  - Suggest repair method
  - Perform repair operation
- The Cleanup tools are available from the Tools menu in GAMBIT.



Semi-Automated Cleanup Tools

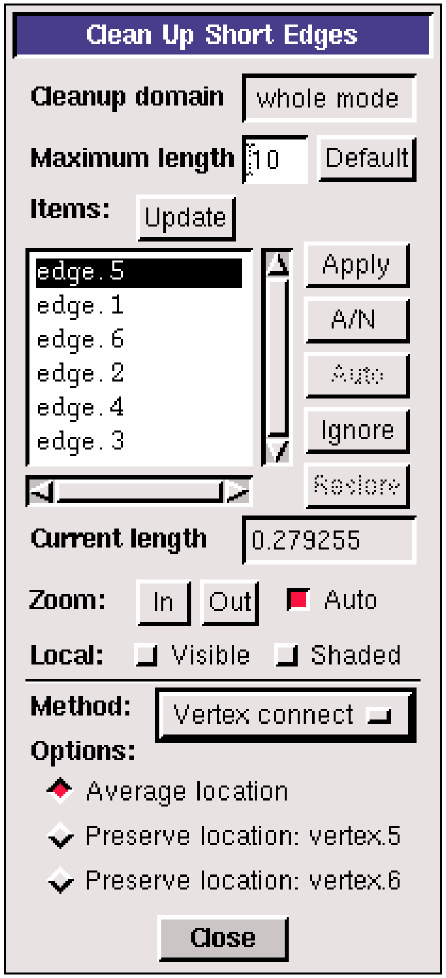


- Quickly identify, zoom in, highlight areas that cause connectivity and mesh quality problems.
  - Appropriate tools to fix problems are given.
  - Graphics color coding set to connectivity based coloring.
  - Graphics window pivot set to mouse.
- Available Cleanup tools:
  - Clean up Short Edges
  - Clean up Holes
  - Clean up Cracks
  - Clean up Sharp Angles
  - Clean up Large Angles
  - Clean up Small Faces
  - Clean up Hard Edges
  - Clean up Fillets
  - Clean up Duplicate Geometry
  - Select Cleanup Domain

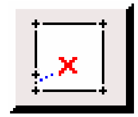


Clean Up Short Edges

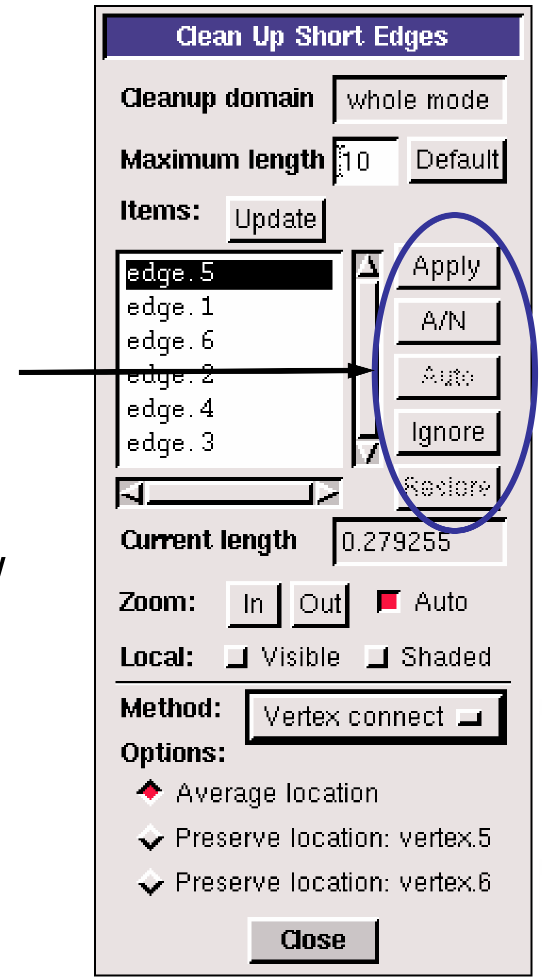
- Tools to identify and highlight the problem spot
  - Cleanup domain
    - Select whole model or group
  - Maximum length: upper limit
    - Default: 10* shortest edge in the Cleanup domain
  - Items List: candidates for cleanup operation based on Cleanup domain and Maximum length
  - Current length: length of currently picked edge
  - Update: updates the Items list
    - Required when Maximum length is modified
  - Zoom
    - In/Out: quick auto zoom in on or from the picked items
  - Auto: automatically zooms in on selected item



Clean Up Short Edges



- Tools to identify and highlight the problem spot
  - Local: current item + all faces connected to it
    - Visible: make everything else invisible
    - Shade: shade the local objects
- Options to Apply Cleanup Tool
  - Apply: applies appropriate fix to selected item
  - A/N: (Apply/Next) applies appropriate fix to selected item and automatically picks the next item in the list. The view is changed.
  - Auto: entire list is processes automatically (only works for the Method: Edge merge)
  - Ignore: removes selected item from list and selects next item
  - Restore: the list is restored

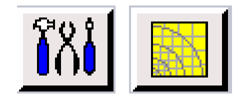


FLUENT
2006 CFD Summit

Meshing Operations

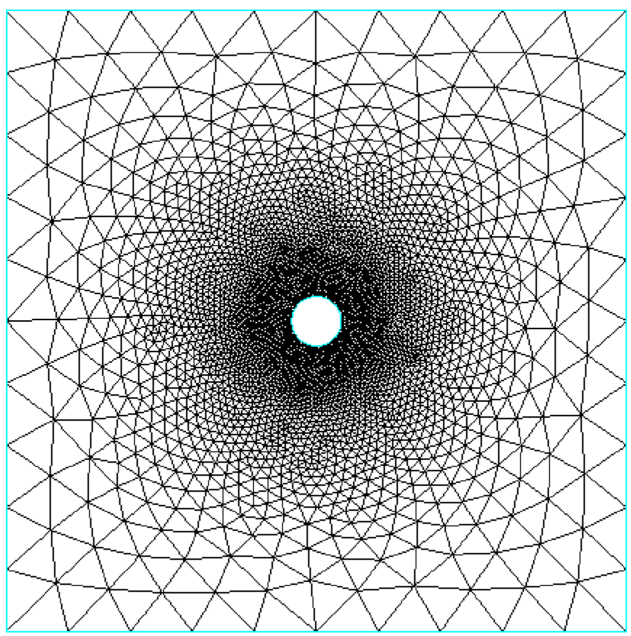
- Size Functions
- Boundary Layers
  - GAMBIT 2.3 Enhancements
- Meshing Tips and Tricks

Size Functions

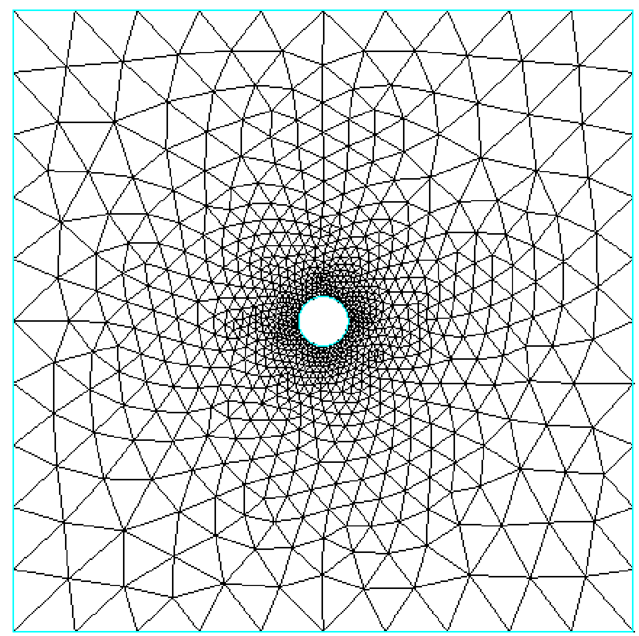


- Control mesh distribution on faces and in volumes
  - Eliminate the need to premesh edges manually
  - Automatically capture important geometry and flow features
  - Ensure high-quality meshes
- Primarily designed for unstructured meshing

Without Size Function



Controlled growth rate with Size Function



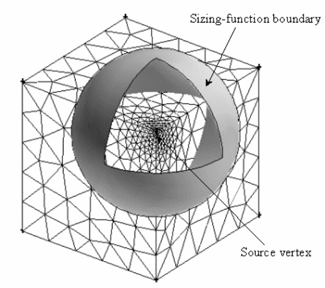
Size Function Types

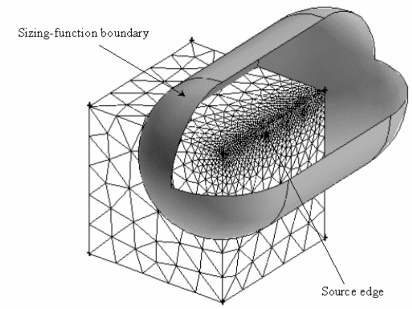
- Fixed
  - Controls mesh element size in a region as a function of distance from a given location
- Curvature
  - Controls mesh element size as a function of geometric curvature of a face or edge and growth of mesh away from source
  - Good for highly curved surfaces or edges
- Proximity
  - Controls the number of of mesh elements in a gap (edge gaps and face gaps) and growth of mesh away from gap
- Meshed
  - Uses existing mesh as initial size on source entity
  - New Size Function type in Gambit 2.2

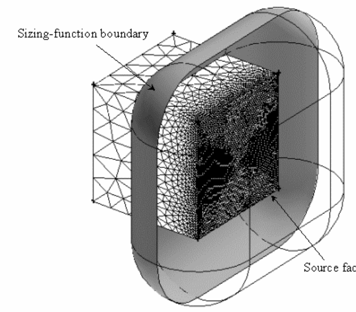
Fixed Size Function

- Requires **Source** specification
  - Source is origin of Size Function, center of region
    - Vertices, edges, faces, and volume can be sources for Fixed Size Function
    - Source entity type determines shape of size function
      - Spherical shape around vertex
      - Cylindrical shape around straight edge
    - Sources can be topologically part of the attachment (**component source**) or independent of attachment (**non-component source**)
    - Sources can be internal or external

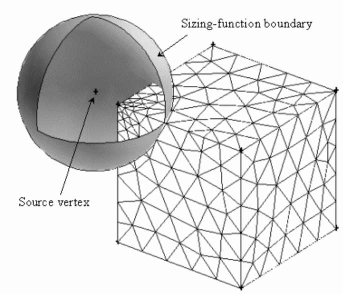
Component sources

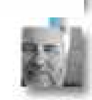






Non-component source



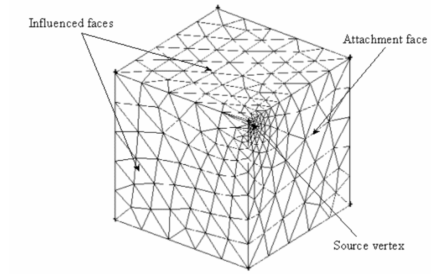


Fixed Size Function

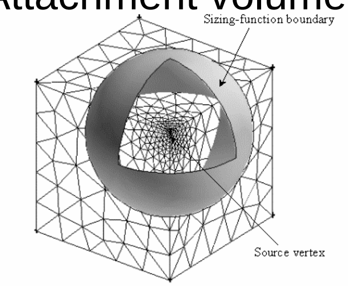
- Requires **Attachment** specification
  - Attachment is the mesh to be affected
    - Fixed Size Function can be attached to edges, faces, and volumes
    - Attachment is ideally the whole model to ensure high quality mesh

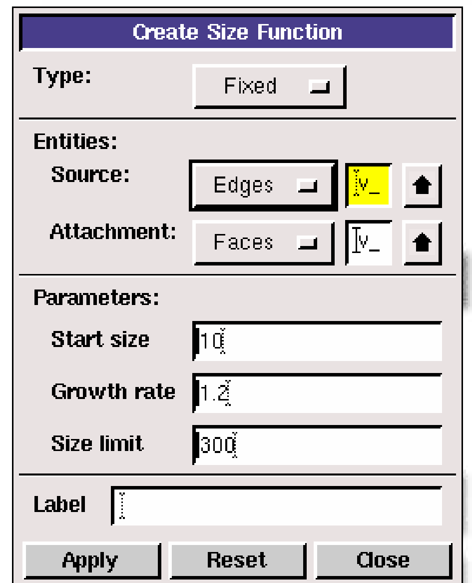
- Parameter specification
  - Start size
    - Mesh element size at source
  - Growth rate
    - Geometric stretching of mesh away from source
    - Cannot be less than 1.0
  - Size limit
    - Maximum allowable mesh size for attachment
    - No further growth if size limit is reached

Attachment face



Attachment volume

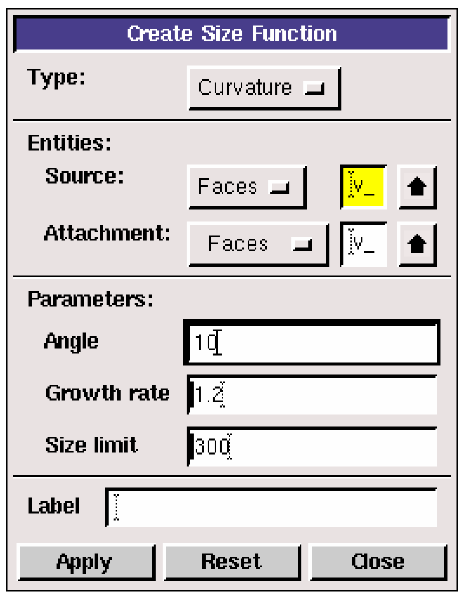




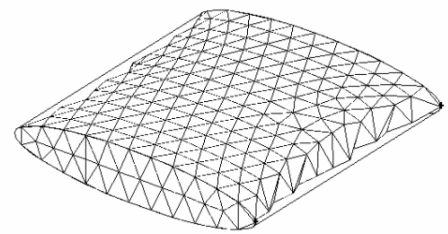
FLUENT
2006 CFD Summit

Curvature Size Function

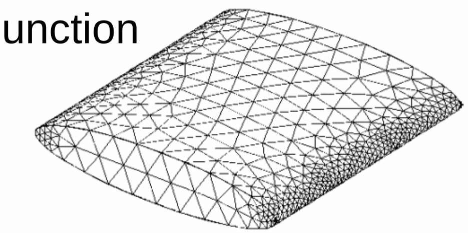
- Source specification
  - Curvature of source entities (edges, faces) used to determine start size
- Parameter specification
  - Angle
    - Maximum allowable angle between any two adjacent face normals on source
    - 5-30 degrees is reasonable value
  - Growth rate
  - Size limit



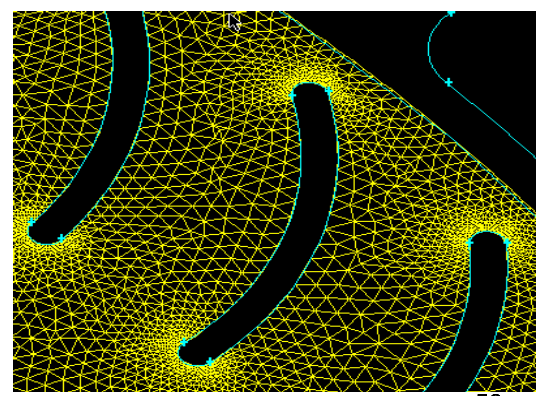
No Size Function



With Curvature Size Function



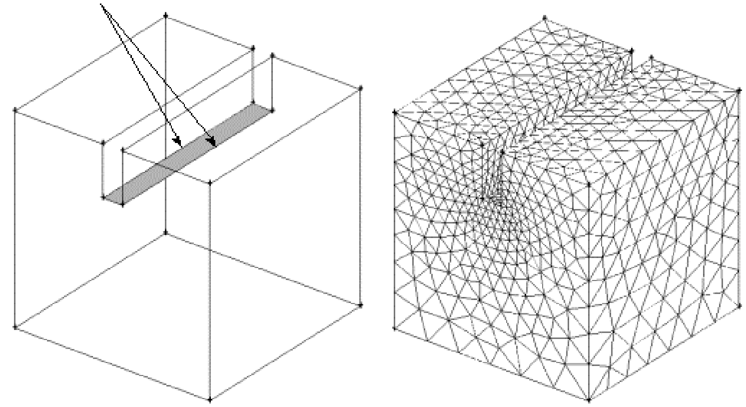
Edges as source entities



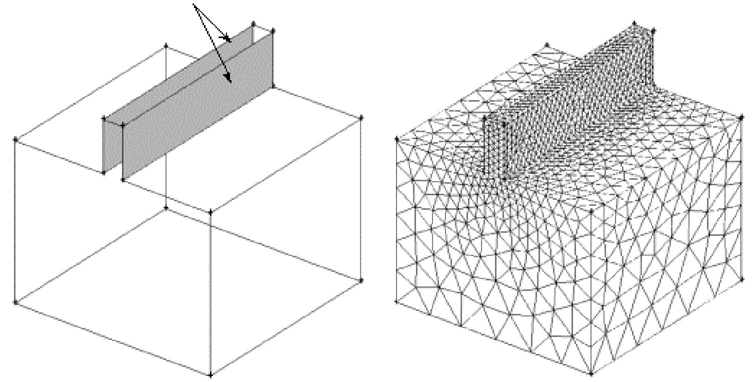
Proximity Size Function

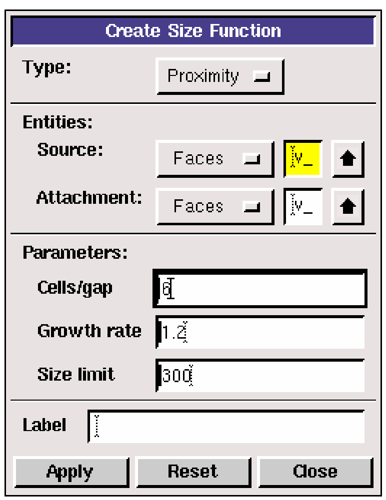
- Source specification
  - Source entities which contain gap
    - Gap is volumetric region between specified faces, or area between opposing edges of specified face
- Parameter specification
  - Cells/gap
    - Minimum number of mesh cells in gap
  - Growth rate
  - Size limit

Gap defined by one source face



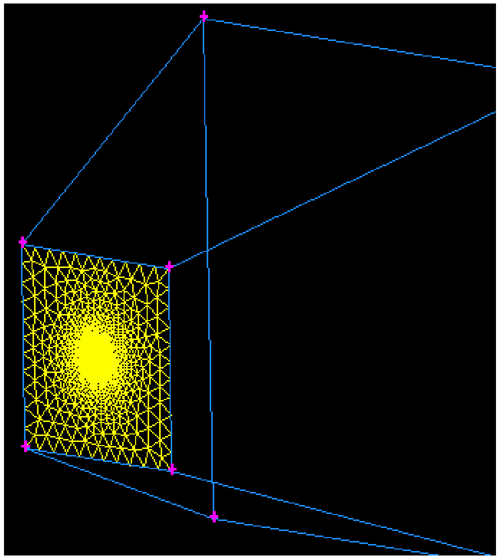
Gap defined by two source faces

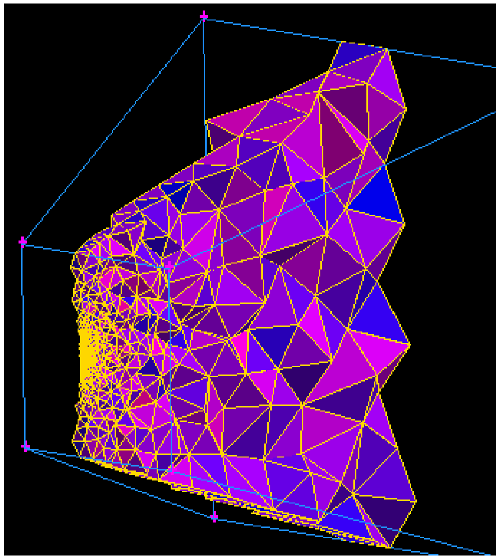


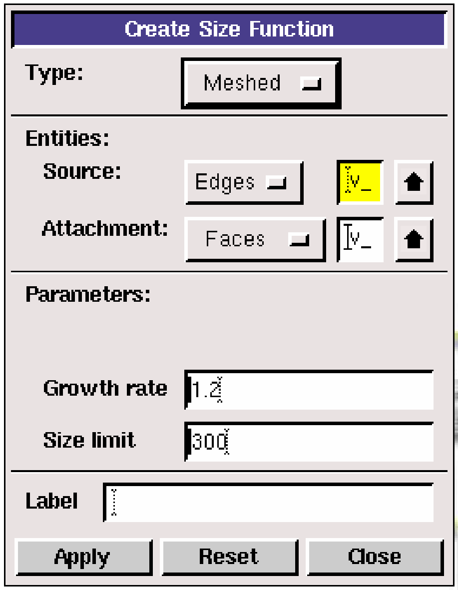


Meshed Size Function

- Ensures that mesh grows controlled away from a premeshed source entities (edges or faces)
  - Very useful after importing mesh
  - Local start size taken from premeshed source entities
- Parameter specification
  - Only growth rate and size limit are required

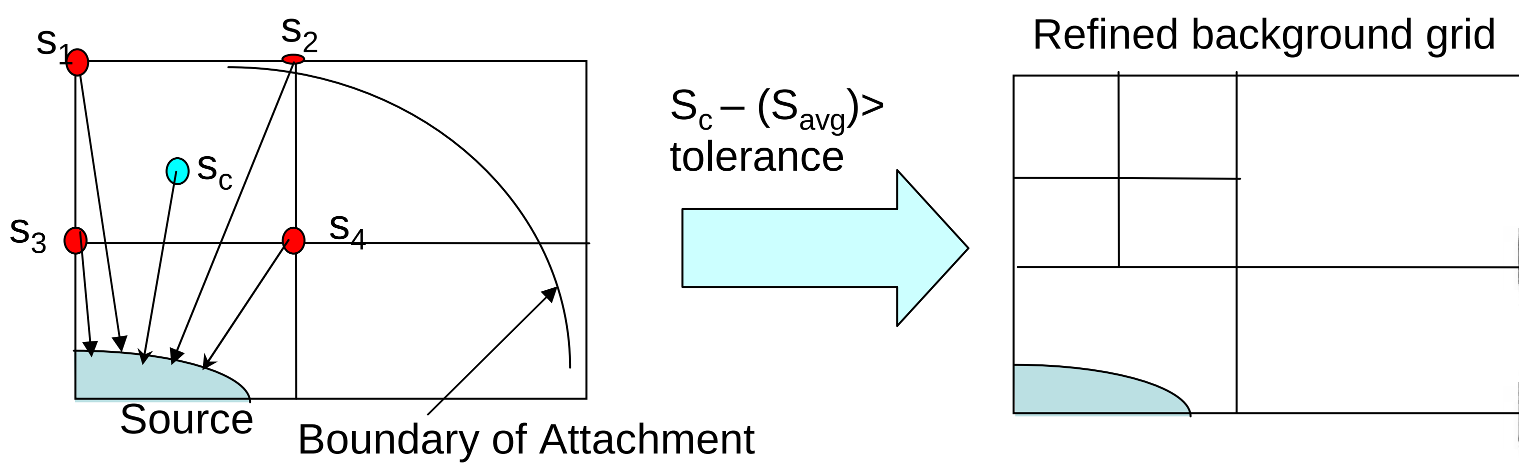






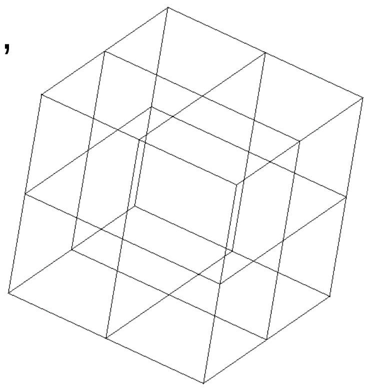
Background Grid Generation

- A set of Cartesian boxes forming a grid that bounds the attachment geometry are generated and refined i.e. split into smaller boxes.
- This successive refinement of the background grid is carried out until a maximum number of levels of refinement (or “tree depth”) are reached or the size variation in all the boxes is less than a specified tolerance limit (“nonlinear error percent”).



Background Grid Defaults

- Use of the background grid default parameters is **key** to obtaining the desired meshes
  - TOOLS.SFUNCTION.BGRID_MAX_TREE_DEPTH controls the maximum refinement of the background grid
    - Increase the default value (16) until no cells hit the tree depth as reported in the transcript window
    - A value of –1 puts no limits on the background tree depth, but makes Size Functions slow for larger models
  - TOOLS.SFUNCTION.NONLINEAR_ERR_PERCENT controls the allowable deviation of the local mesh from the prescribed mesh size
    - Default is 25%, can vary between 3 and 25%
    - Number of cells above the prescribed tolerance are reported in the transcript window
  - TOOLS.SFUNCTION.REPORT_BGRID_INFO = 1 turns reporting in the transcript window on (on by default)



Background Grid

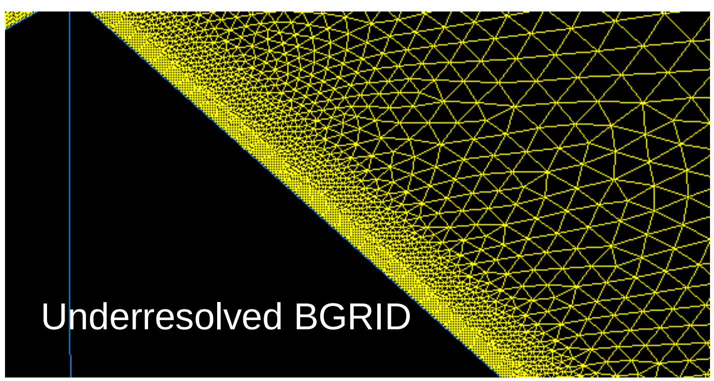
- Use the Size Function reporting in the Transcript window
  - If the mesh is noticeably changing as BGRID_MAX_TREE_DEPTH is increased, it is a sign that too many background grid cells are hitting the limit.

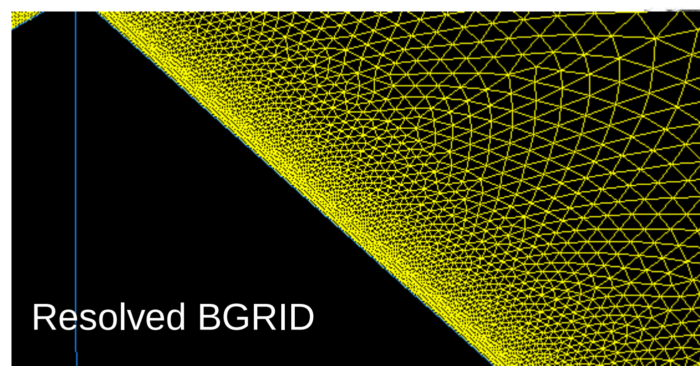
```
Command> default set "TOOLS.SFUNCTION.BGRID_MAX_TREE_DEPTH" numeric 16
The start date and time = Thu Apr 24 09:23:06 2003
The finish date and time = Thu Apr 24 09:23:29 2003
CPU time used to generate background grids (sec.) = 22.76
Maximum background grid level = 16
Total number of background grids = 3321
Number of cells hitting tree depth = 601
% of cells hitting tree depth = 18.10 %
Number of cells below specified tolerance = 2720
% of cells below specified tolerance = 81.90 %
Worst tolerance % of BG cells = 60

Command> default set "TOOLS.SFUNCTION.BGRID_MAX_TREE_DEPTH" numeric 22
The start date and time = Thu Apr 24 09:27:12 2003
The finish date and time = Thu Apr 24 09:27:45 2003
CPU time used to generate background grids (sec.) = 33.32
Maximum background grid level = 19
Total number of background grids = 4867
Number of cells hitting tree depth = 0
% of cells hitting tree depth = 0.00 %
Number of cells below specified tolerance = 4867
% of cells below specified tolerance = 100.00 %
```

Background grid level reached maximum value specified

Size Function not sufficiently resolved

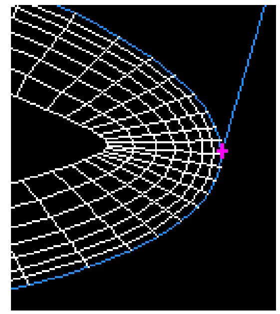
Ideal background grid

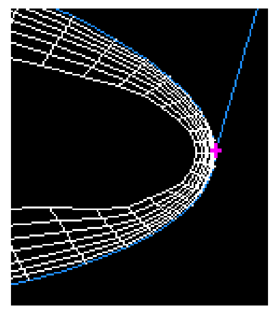


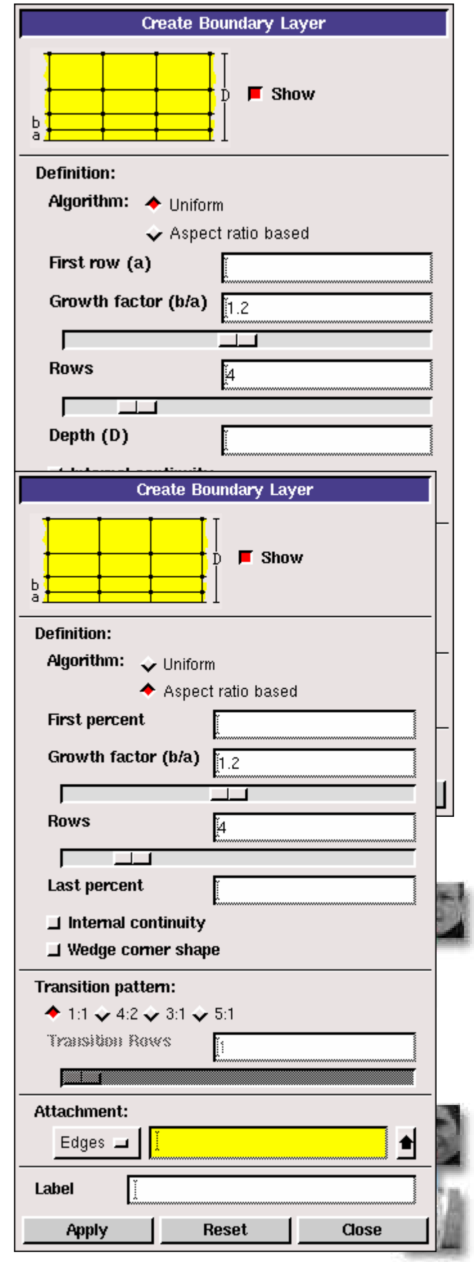


Boundary Layers

- Produce high quality cells near boundary
  - Attachment to edges for 2D BL
  - Attachment to faces for 3D BL
- Uniform or aspect ratio based BL
  - Uniform
    - First row height as parameter
    - Difficulties on highly curved geometry
  - Aspect ratio based
    - Aspect ratio of first cell is parameter
    - Does well in concave corners
    - Cannot directly control first cell height

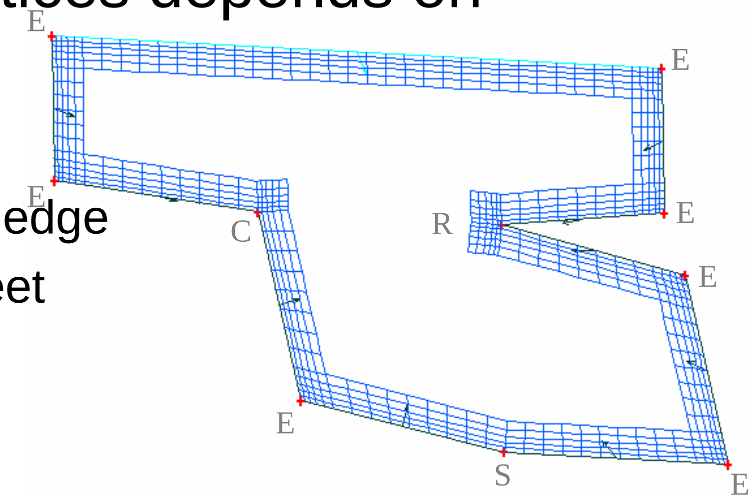




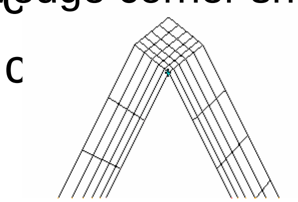


2D Boundary Layers

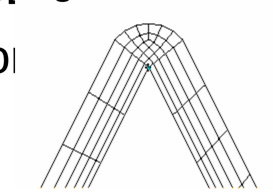
- Boundary layer shape near vertices depends on vertex type
  - End (E) type
    - BL mesh butts up against adjoining edge
    - Overlapped mesh where two BL meet
  - Side (S) type
    - Angle at vertex is bisected
    - Continuous BL
    - ‘Internal continuity’ enforced by manually changing vertex type to S
  - ‘Wedge corner shape’ option for corner (C) and reversal (R) vertices
    - Wedge c … applies to 2D B…
    - Can be c … Cooper tool fo… L



Wedge corner shape off



Wedge corner shape on

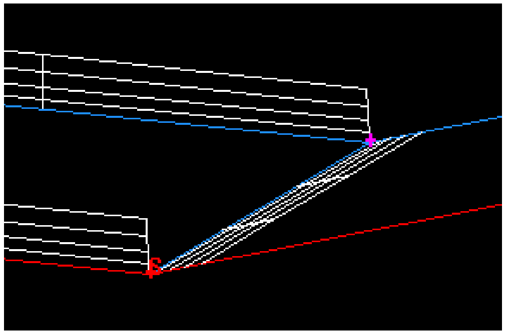


FLUENT 2006 CFD Summit

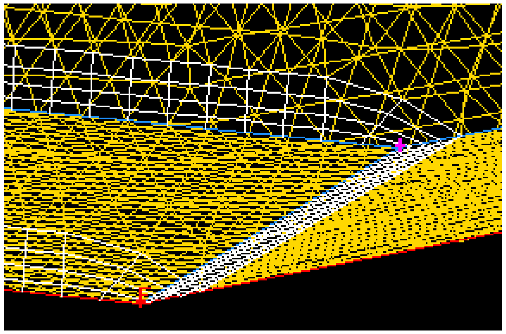
3D Boundary Layers

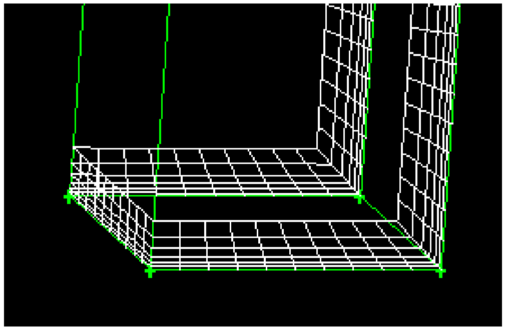
- Internal continuity
  - Allows boundary layers to be formed without crossover region
- Imprinting on adjacent face
  - Single default MESH.VERTEX.MAX_END_ANGLE (=120º by default) for consistent imprinting of 2D and 3D BL

Gambit 2.1

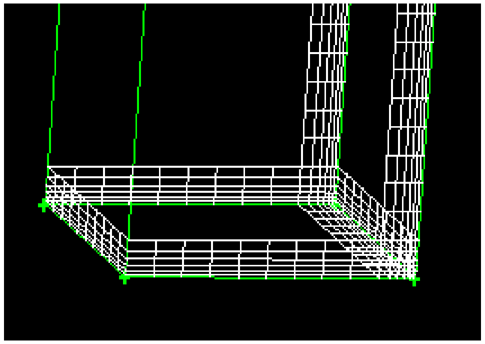


Gambit 2.2



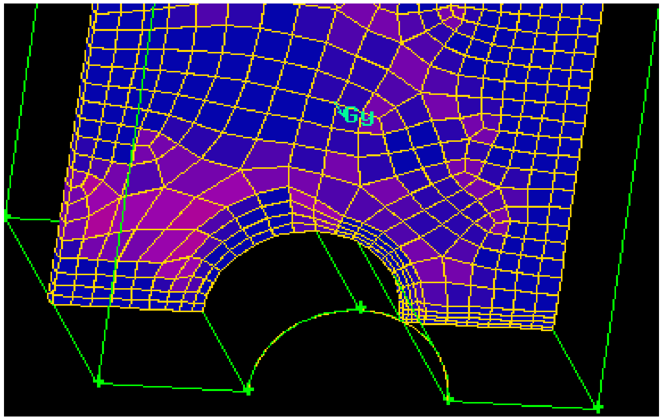


Internal continuity ‘on’

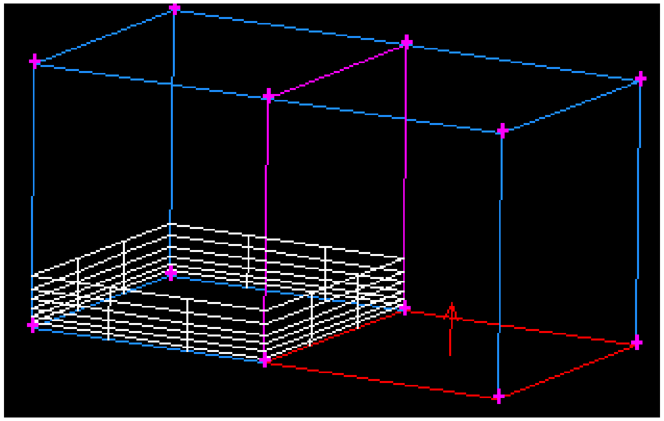


Internal continuity ‘off’

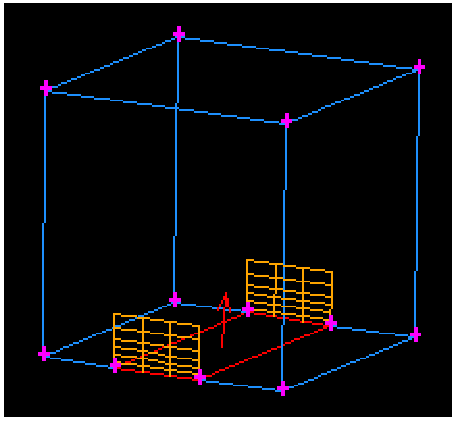
3D Boundary Layers



- Different 3D BL settings allowed on adjacent surfaces.



- Imprinting of 3D BL fails if adjacent volume has already BL attached
  - Modify existing BL instead

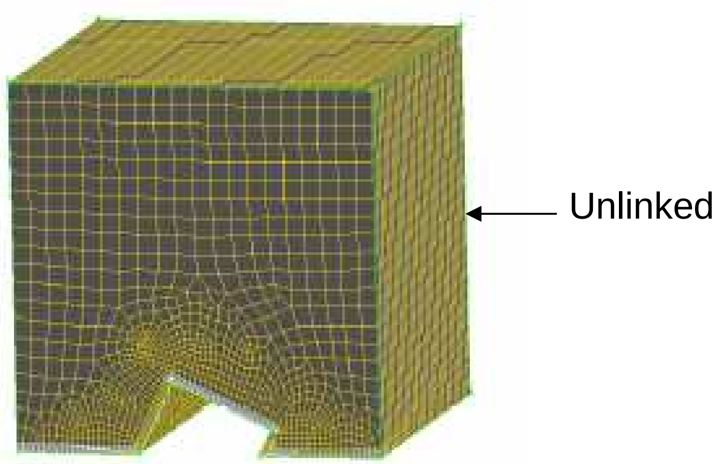


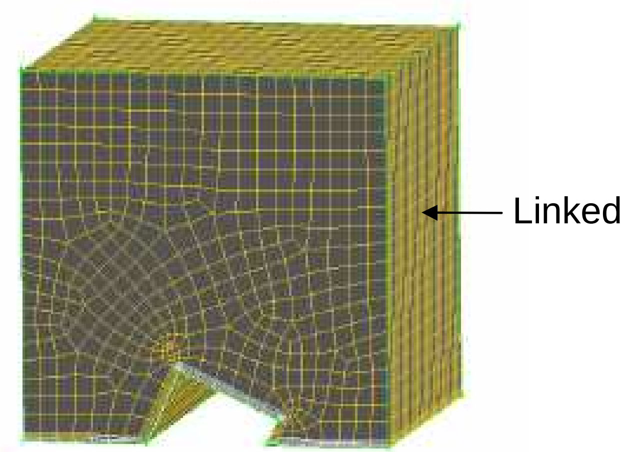
- Dangling 3D BL are not supported

FLUENT
2006 CFD Summit

Boundary Layers and Cooper Meshing

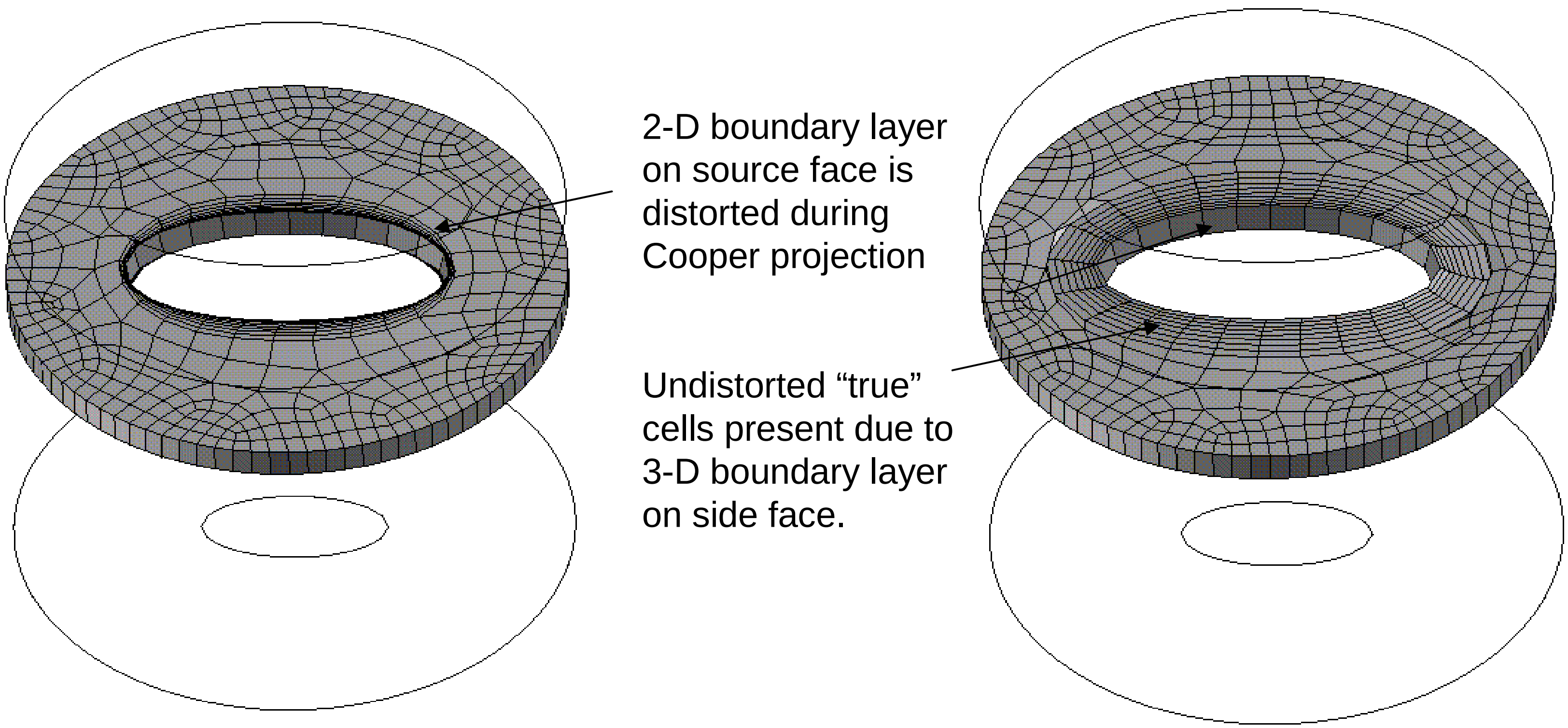
- Both 2-D and 3-D boundary layers can be used with the Cooper Meshing Scheme:
  - If a source face has a 2-D boundary layer attachment on its edges, (or 3-D imprinted boundary layer), the Cooper tool will generally create corresponding elements in the volume.
  - 2-D “linked” boundary layers can be used when source faces are linked (1-to-1 relationship).





Boundary Layers and Cooper Meshing

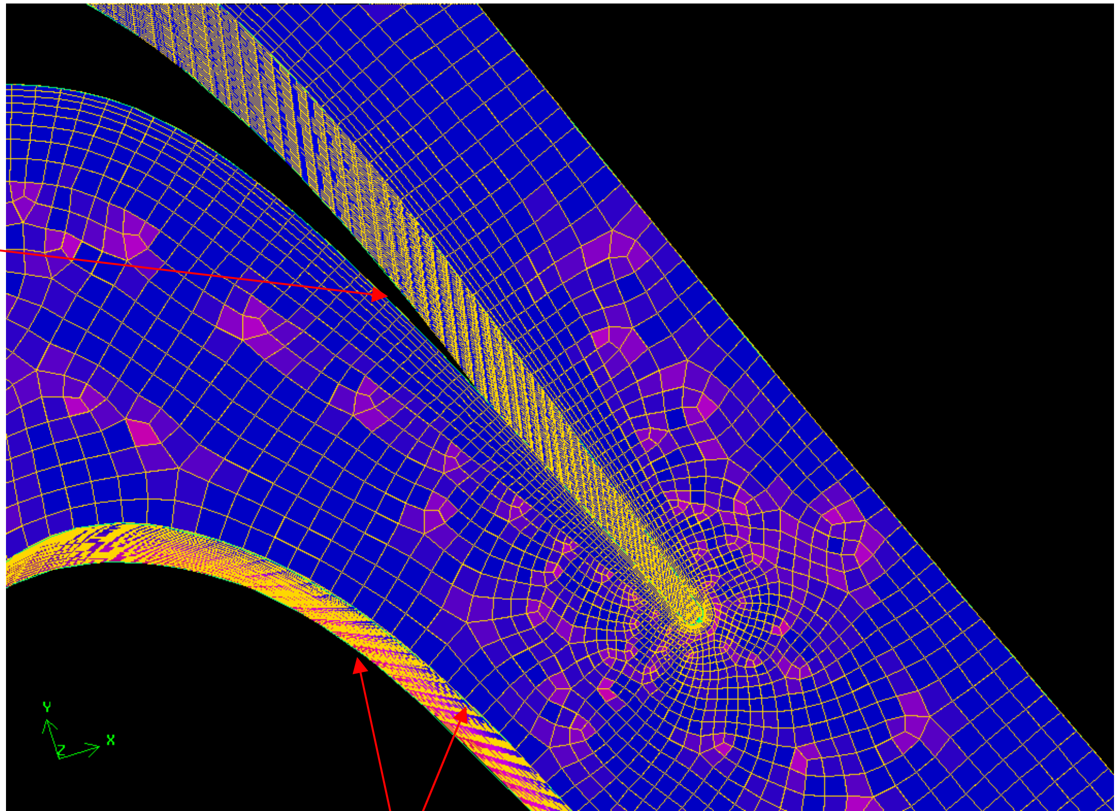
- 3-D boundary layers are advantageous where there is a significant twist in the geometry in the Cooper direction to avoid large change in cell size and thickness due the mesh projection along the Cooper direction.



Boundary Layers and Cooper Meshing

- Aspect Ratio based boundary layer can be used to ensure square cells at the outer layer.

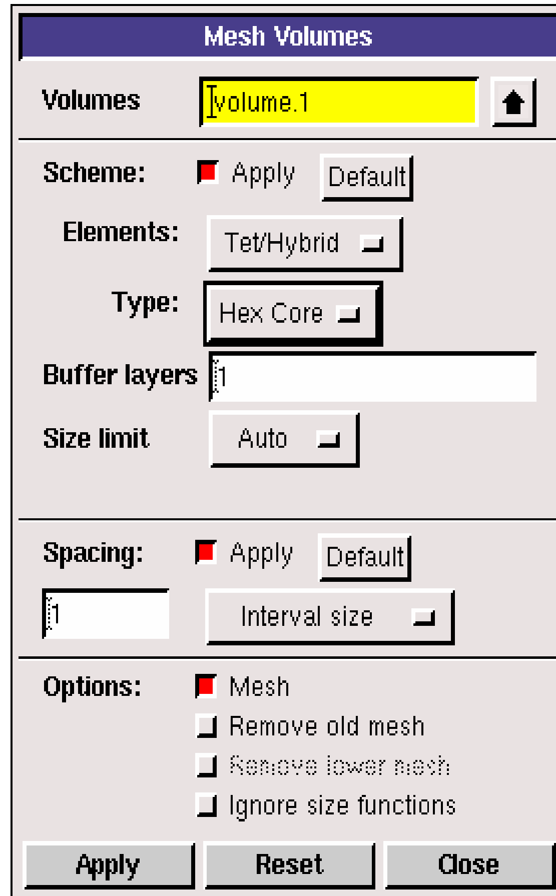
Curvature size function is used to dictate the face mesh around curved boundaries.



Source faces are soft linked for 2-D Boundary layers.

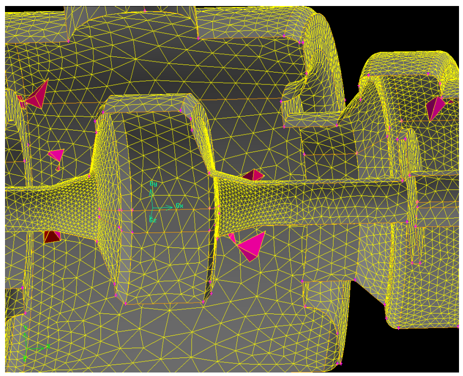
Enhancements to Hexcore Capabilities in GAMBIT 2.3

- HexCore Meshing
  - New meshing parameters
    - Buffer Layers
    - Size Limit
  - Size Function is not required anymore.
    - Leads to faster meshing.
  - A Non-conformal mesh is now produced automatically.
    - Use old HexCore with FIDAP by setting HEXCORE_METHOD = 0 and QUAD_SURFACE_SPLIT = 0.
  - Optimal mesh with Boundary Layers.
    - Intersection with boundary layer cap instead of boundary.

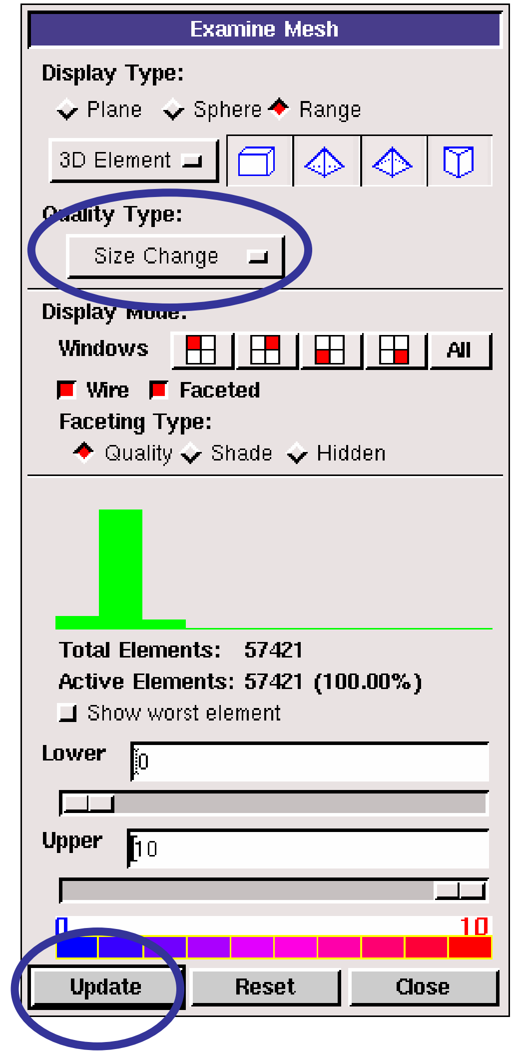


Enhancements to Examine Mesh Form in GAMBIT 2.3

- A new “Size Change“ quality option has been added.
  - Find large size jumps in the mesh between neighbouring cells.

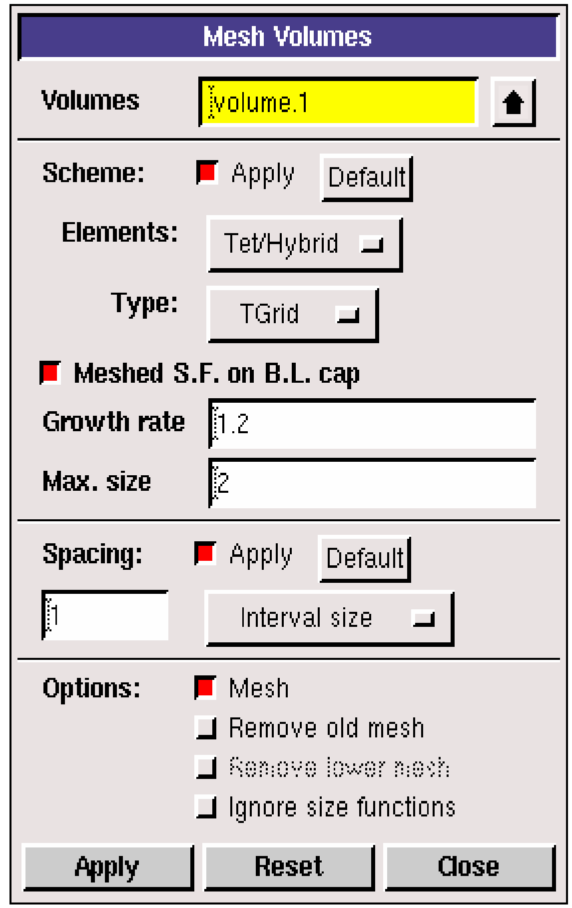


- An Update button has been added to delay graphics updates until the user clicks the button.
  - Avoids graphics problems when examining large meshes using selecting multiple options on the form.



GAMBIT 2.3: Meshed Size Function from Boundary Layer Cap

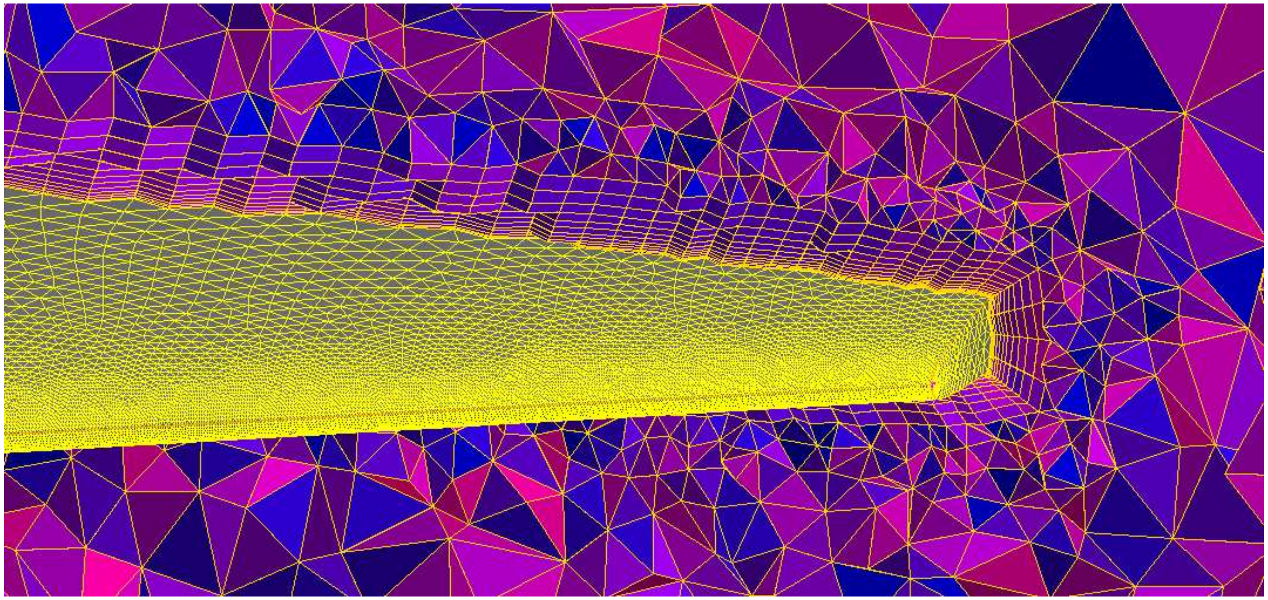
- Meshed Size Function starting from Boundary Layer cap is now available in the Mesh Volume form for tetrahedral meshing (Tgrid scheme).
  - Parameters specified
    - Growth rate
    - Max size
  - Improves size transition between Boundary layer and Volume mesh.
  - Useful for external aerodynamics applications.
  - Ignores all other size functions.



FLUENT
2006 CFD Summit

Example: 3D Wing Profile with 12 boundary layers

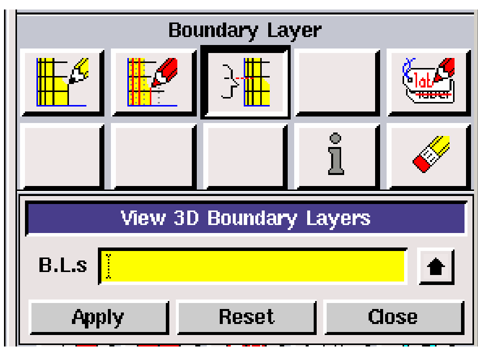
- A Meshed Size Function attached to the wing surface gives 100 highly skewed tets, due to a size jump between boundary layer and tet mesh.
- The meshed size function attached to boundary layer cap yields only one highly skewed element with a smooth transition in cell size.

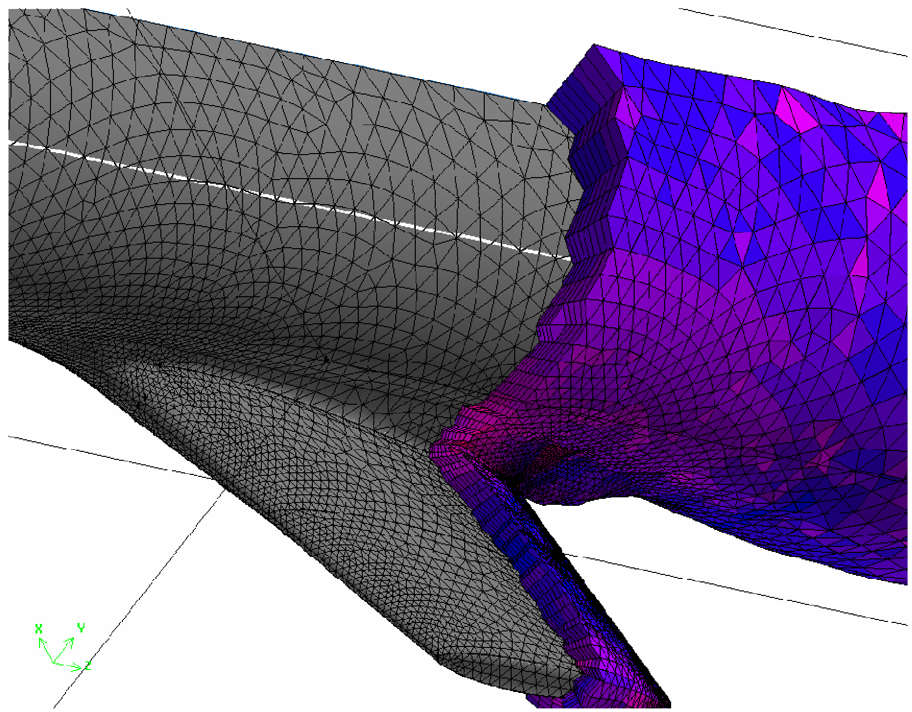


FLUENT
2006 CFD Summit

GAMBIT 2.3: View 3D Boundary Layers Capability

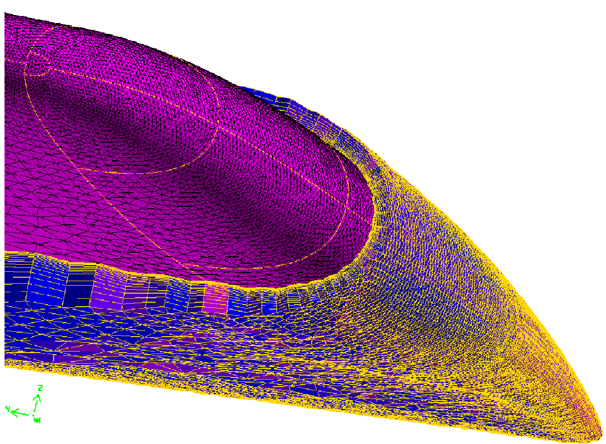
- The new View 3D Boundary Layers allows users to examine 3D boundary layer mesh prior to volume meshing.
    - Resolve quality issues
    - Resolve tet-failure

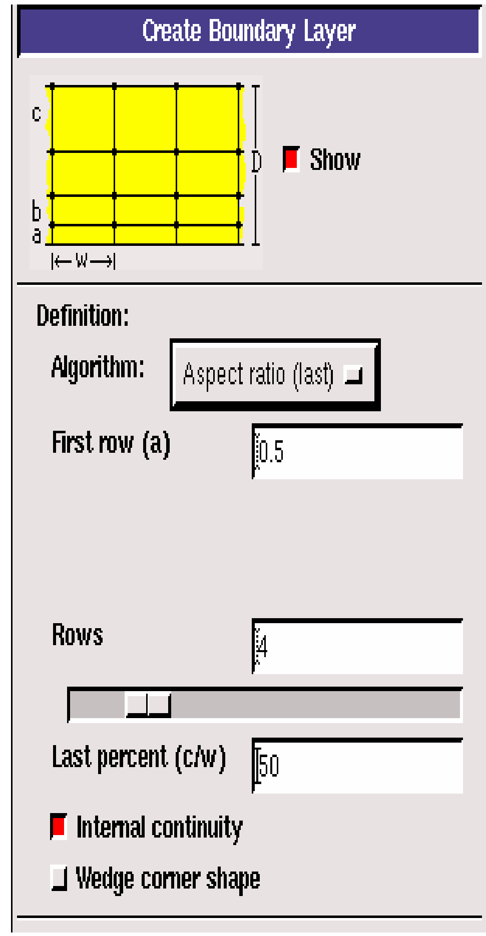




GAMBIT 2.3: Last Aspect Ratio Boundary Layer Type

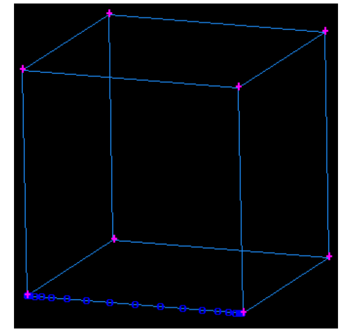
- The new Last Aspect Ratio Boundary Layer type allows the growth of high quality cells (“square” cells) in the last layer of a 3D boundary layer.
  - Specify fixed first height, number of rows and last aspect ratio.
  - Critical for external aerodynamics applications.

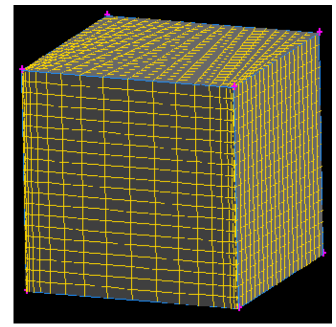




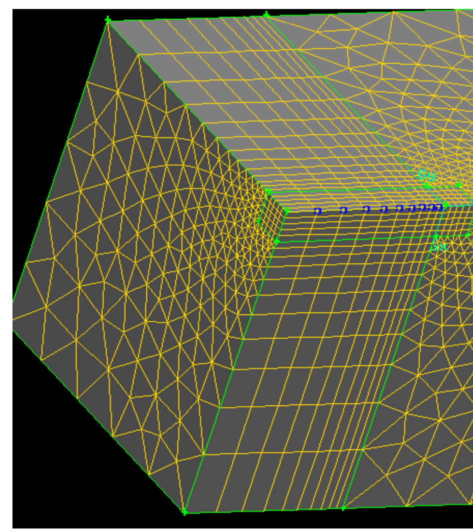
Misc. Map Meshing Tools

- Auto-match distribution of premeshed edges on opposite edges
  - MESH.MAP.MATCH_PREMESHED_EDGE_GRADING = 1
  - Auto-match doesn't propagate to edges of adjacent faces when map-meshing volumes



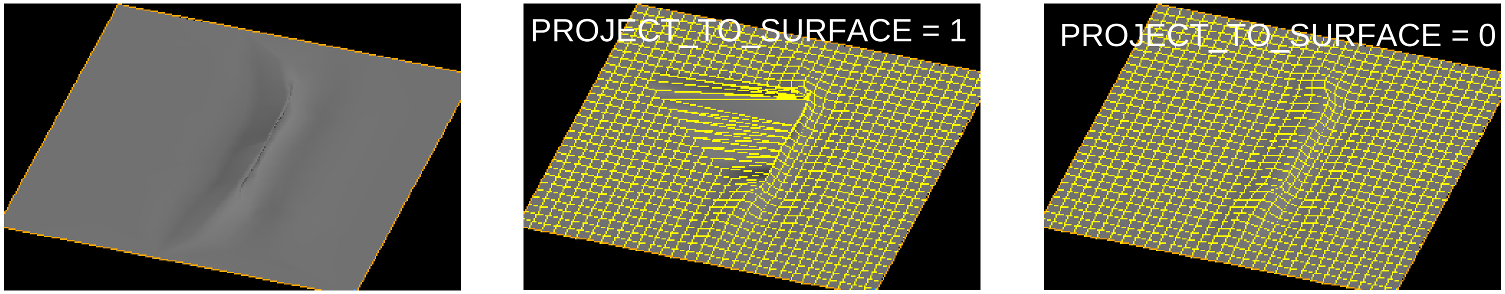


- Sweep and revolve of faces generates 3D meshes from 2D meshes
  - Use 'With mesh' option in sweep and revolve GUI
  - Uses grading and spacing of premeshed sweep path
  - Same option when sweeping or revolving edges into faces

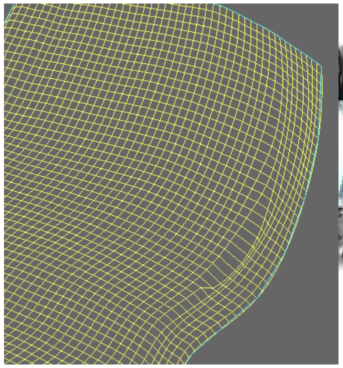


Meshing Tips

- Surface meshing done on faceted face representation by default
  - MESH.FACE.EXACT_MESH_EVALS = 0
- Final exact projection from facets to surface by default
  - MESH.FACE.PROJECT_TO_SURFACE = 1
  - Turn final projection off and leave mesh on facets if projection generates skewed or degenerated mesh on bad/corrupt surfaces



- Poor mesh by mapper on highly curved surfaces
  - Projection to the surface distorts the mesh
  - Smooth the face mesh using 'Winslow' scheme (only available for quad elements)



FLUENT
2006 CFD Summit

Meshing Strategies

- Meshing Methodology
- Selecting a Meshing Strategy
- Meshing Strategies
- Meshing for Quality
- Meshing for Physics

Meshing Strategies

- A number of different meshing strategies can be used for dealing with “real-world” geometries:
  - Hex dominant mesh using geometry decomposition
  - Tet dominant mesh
  - HexCore mesh
- Each approach has a trade-off between computational cost, mesh size, quality and resolving physics and the time and effort required for meshing.
  - Mesh generation can take more than 50% of total analysis time.
- Selecting the right meshing strategy is a critical task.

Selecting A Meshing Strategy

- The best strategy for dealing with complex geometry depends on
  - Time available
    - <u>Faster</u> tet-dominant mesh vs. <u>crafted</u> hex/hybrid mesh with lower cell count.
  - Desired cell count
    - Low cell count for resolving overall flow features vs. high cell count for greater detail
    - HexCore mesh vs. tet-dominant mesh.
  - Desired mesh quality
    - What is the maximum skewness and aspect ratio you can tolerate?
  - Physics
    - Flow features, resolving turbulence

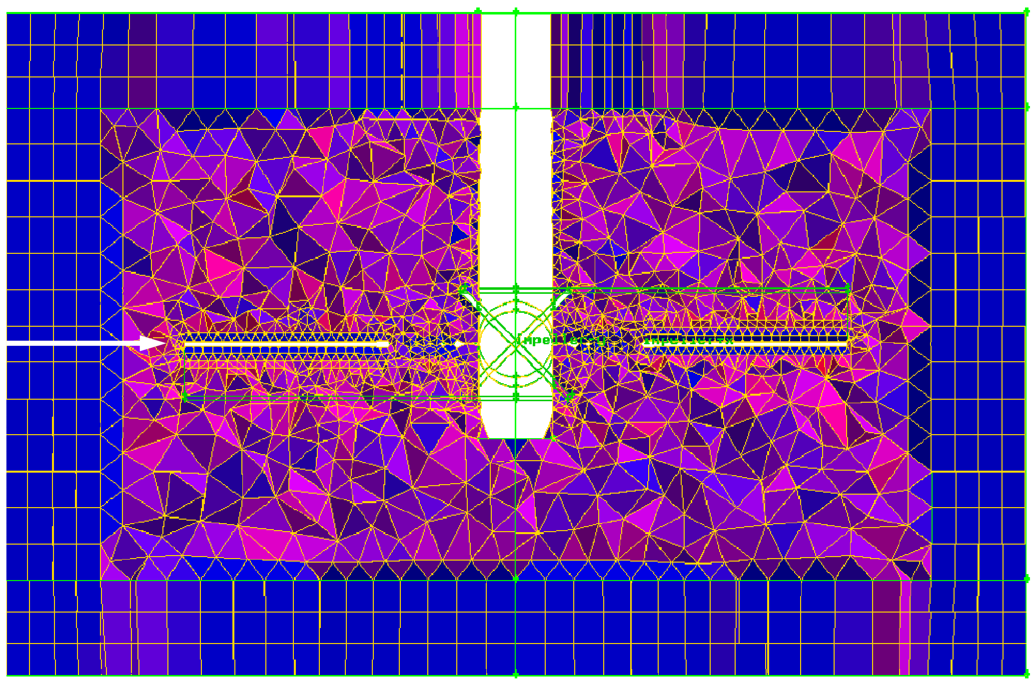
Hex-Dominant Meshing

- The geometry is decomposed into multiple meshable volumes using Boolean operations and splits.
- The following meshing tools are used to create a hex mesh:
  - Map
  - Submap
  - Tet-primitive
  - Cooper
- A tetrahedral (tet) may be created for a complex sub-volume.
  - E.g.: The sub-volume containing the impellers in a mixing tank.
- This works well for prismatic and nearly prismatic geometries.
  - Advantage: Reduced cell count with higher mesh quality translating to faster turnaround time on simulation.
  - Disadvantage: The decomposition may be time consuming for complex geometries.

Mixing Tank

- Mixing tank:
  - The mixing tank is decomposed and a hex mesh is created everywhere, with tets in the impeller volume.

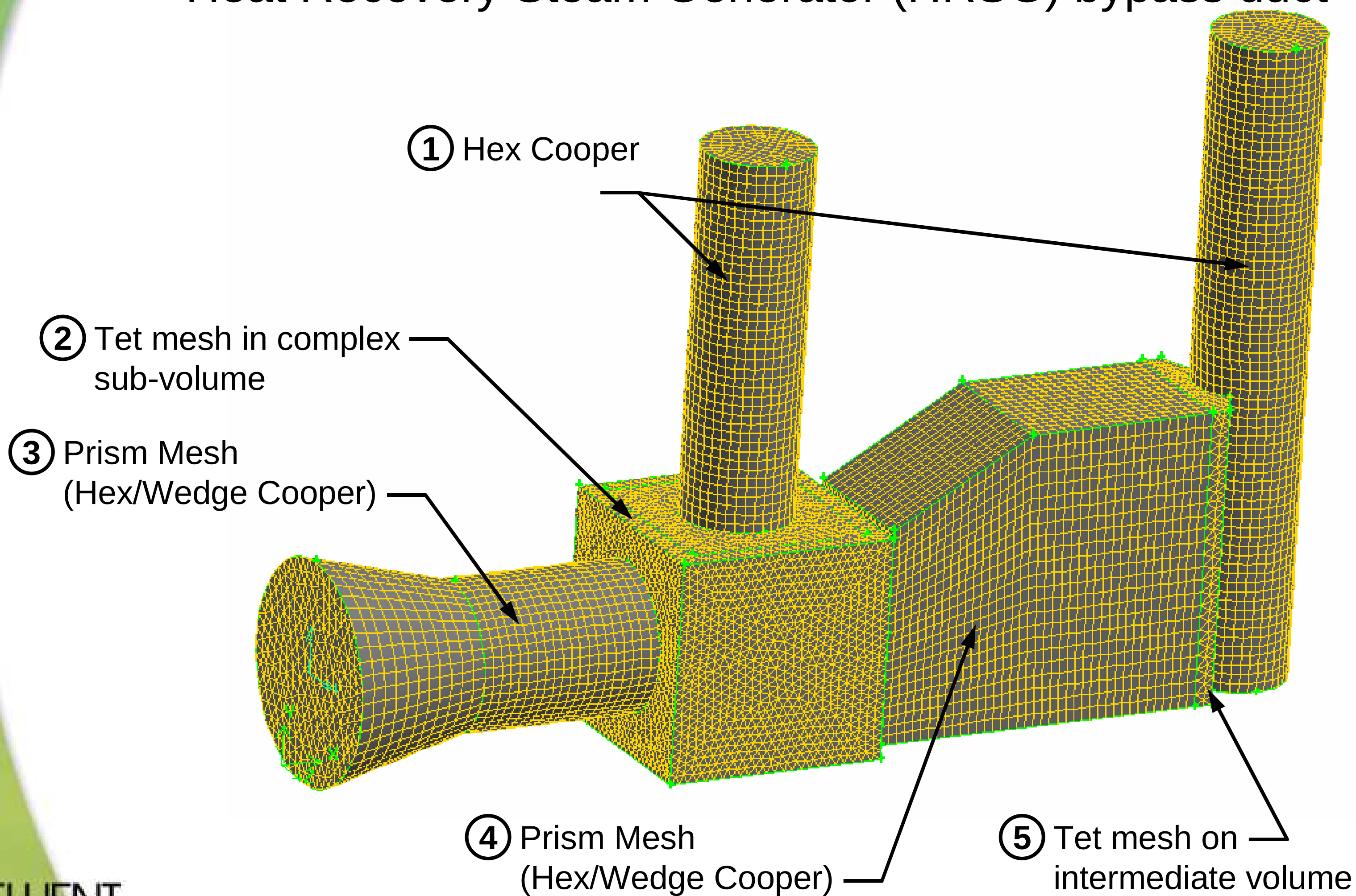
Tets in impeller zone



FLUENT
2006 CFD Summit

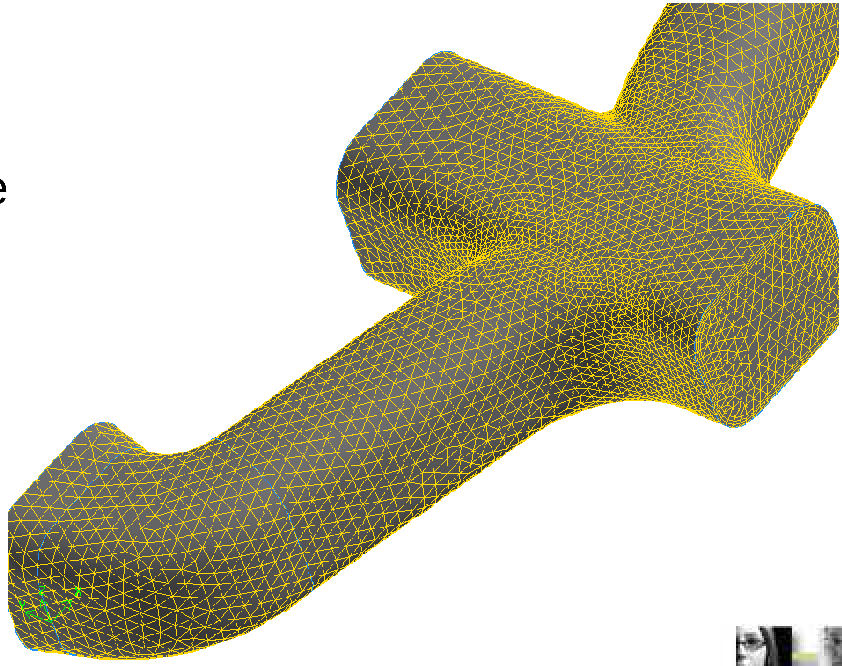
Other Examples

- Heat Recovery Steam Generator (HRSG) bypass duct



Tet-Dominant mesh

- Mostly tetrahedral mesh with some areas containing pyramid, prism and hex elements.
  - Boundary layer prisms/hexes are grown from surfaces where wall effects (boundary layer) are important.
- Size functions attached to the flow volume can be used to create grading in the surface/volume mesh.
  - Global size functions to resolve curvature and proximity.
  - Local size functions for refinement in some areas.

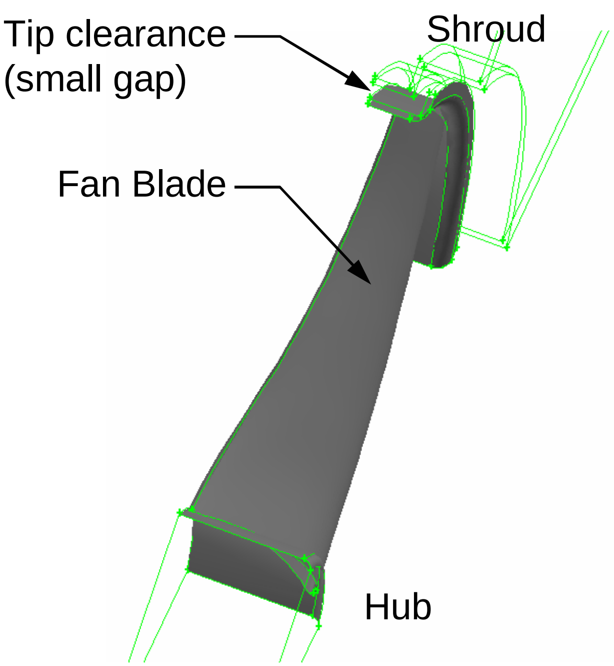


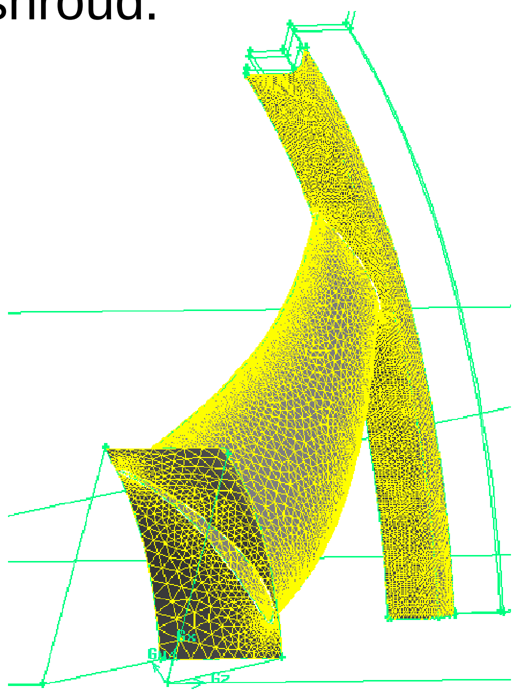
Manifold Geometry with tet-dominant mesh using sizing functions and boundary layers

FLUENT
2006 CFD Summit

Axial Fan Blade

- A curvature size function was used to resolve the blade geometry.
  - Boundary layer attached to the blade to resolve near-wall effects.
- A proximity size function was used to resolve the tip clearance gap.
- Fixed size functions were used to grow the surface mesh away from the blade at the hub and shroud.

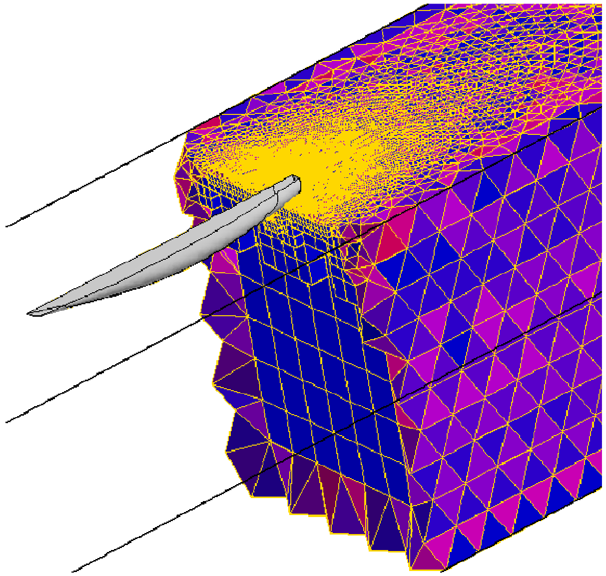




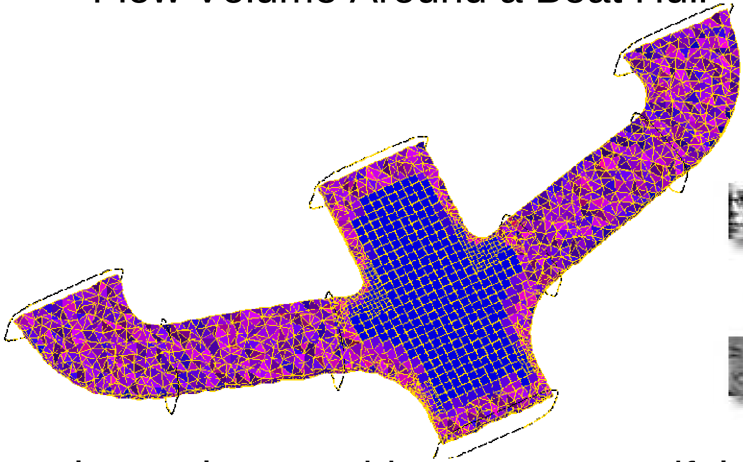
FLUENT 2006 CFD Summit

HexCore Mesh

- HexCore meshing can dramatically lower the mesh count and improve overall quality on volumes with complexity at the surfaces.
- The number of offset layers, as well as size functions, can be adjusted to minimize the high skewness tets resulting from any narrow gaps between the boundary and the HexCore.
- HexCore may not be compatible with some physical models and have size jumps between the tets and hex core.



Flow Volume Around a Boat Hull

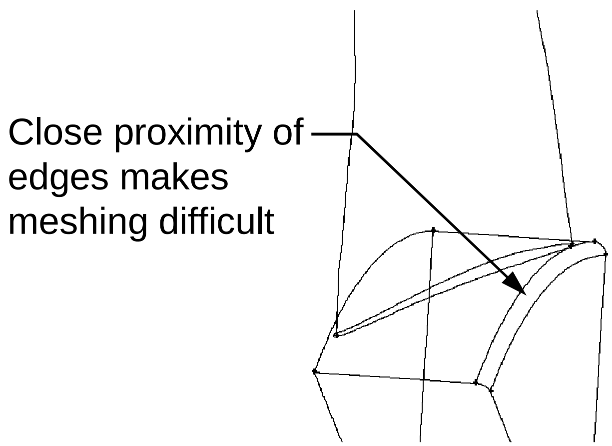


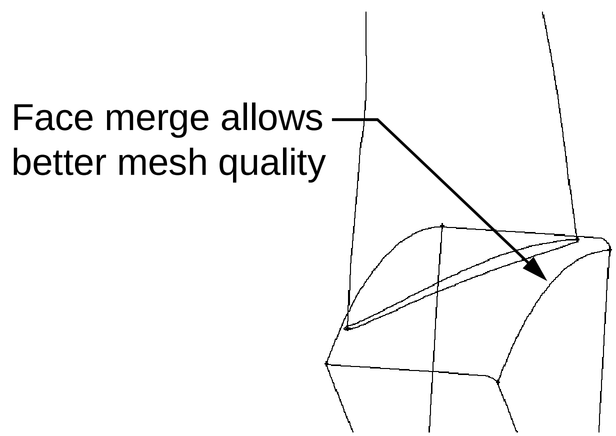
Flow Volume Inside an Auto Manifold

Meshing for Quality

- Mesh quality affects the face flux calculations between cells and hence directly impacts the accuracy of the solution and ease of convergence.
  - Avoid degenerate elements (skewness ~ 1) or high aspect ratio (~5) in the flow volume and high aspect ratio (~100) in the boundary layer.

- Good mesh quality depends on:
  - Clean surface geometry and volumes not having slivers and small features close to complex surfaces.
  - Resolving geometric features well (gaps, curvature, sharp angles etc.)
  - Creating a good surface mesh with appropriate sizing function growth rate and size limit, edge grading, spacing etc.

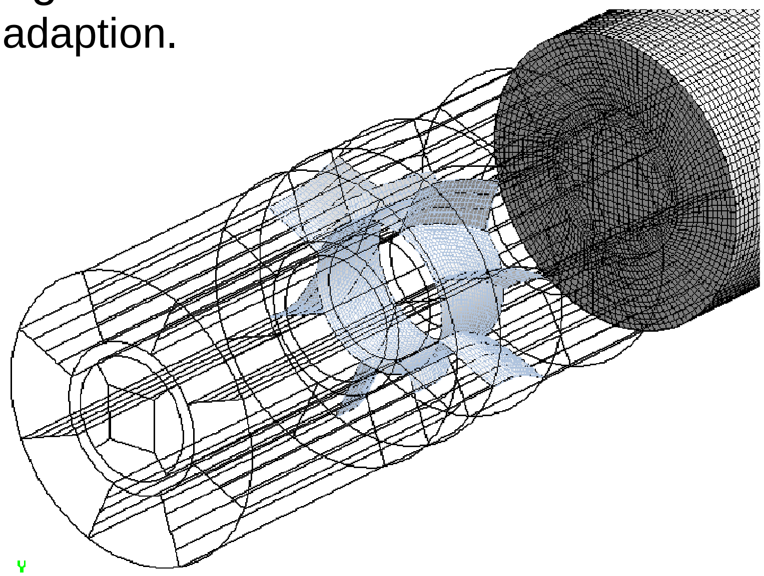




Meshing for Physics

- Complex physics on complex geometry requires greater care in estimating the lowest mesh size required to resolve the physics and grading mesh away from that size.
  - Some flow features can be calculated and resolved with appropriate mesh: jets, wall boundary layers, smallest eddies in LES
  - Some flow features are functions of boundary conditions (recirculations, shocks, vortex lines etc.) and cannot be fully anticipated in size, location and shape.
- The objective is to get a "good enough" or "reasonably good" mesh and then refine it further using FLUENT results.
  - Remeshing in GAMBIT or adaption.

Hex mesh used for LES modeling of confined swirling coaxial jets expanding into a pipe.



FLUENT
2006 CFD Summit

Summary and Conclusions

- GAMBIT provides a comprehensive set of tools for preprocessing:
    - CAD Import
    - Geometry creation
    - Cleanup tools: virtual operations and semi-automated cleanup tools.
    - Mesh control tools: size functions and boundary layers.

- New capabilities have been added to GAMBIT 2.3 to improve robustness and meshing capabilities:
    - Advanced covering
    - Geometry creation and plug-in tools
    - Boundary layer enhancements: New boundary layer type, Meshed size function from boundary layer, View 3D boundary layer
    - Size change quality measure

- Guidelines have been provided for selecting the right meshing strategy for your geometry and modeling problem.