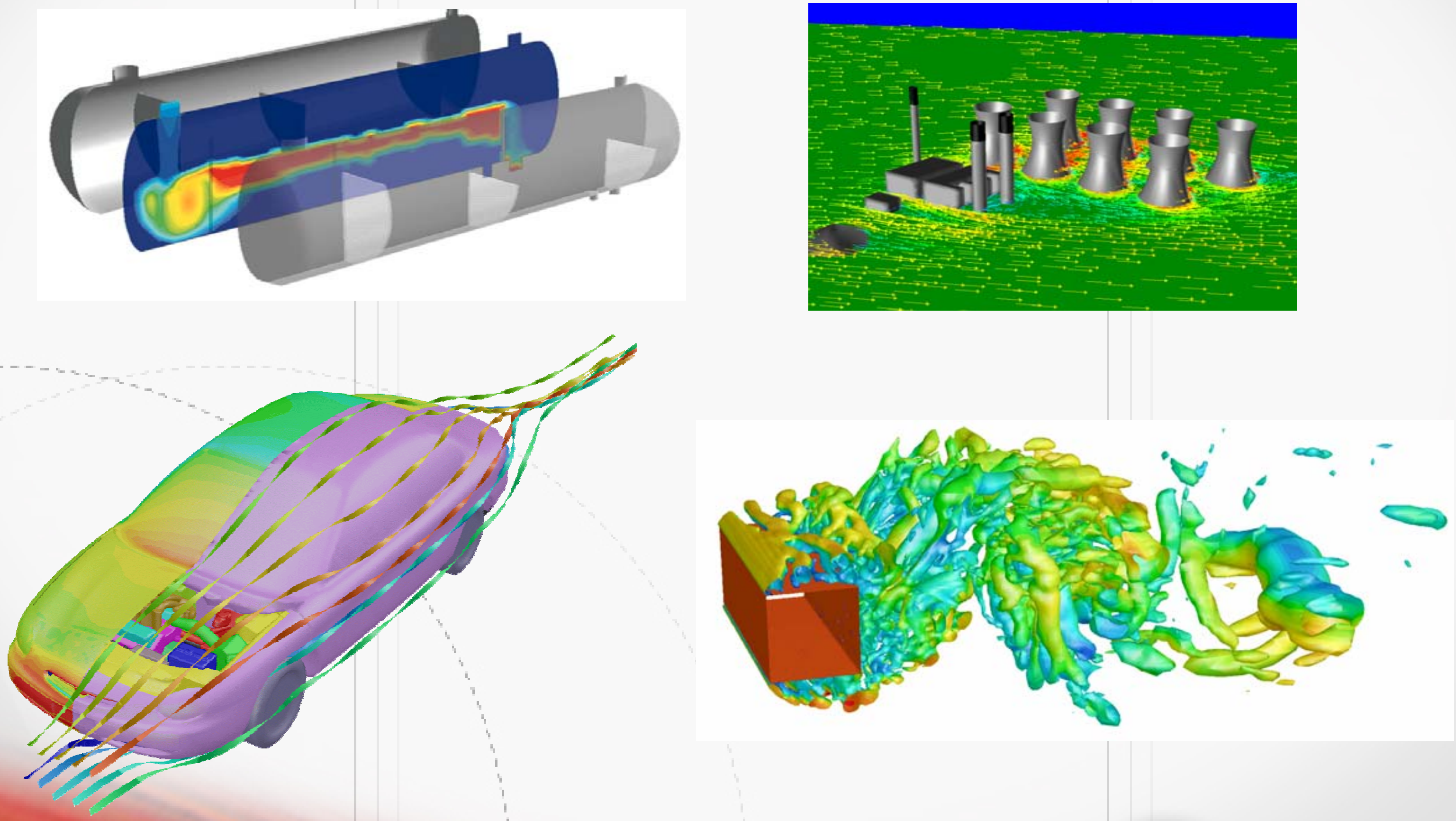
Fluent Inc.



FLUENT
The Right Answer in CFD

Historical Background

- The original FLUENT software, TEMPEST, was developed at the University of Sheffield, UK in the 1970s
- 1983 title in FLUENT was transferred to a New Hampshire company called ‘Creare’
- Office in the UK from 1983

Historical Background

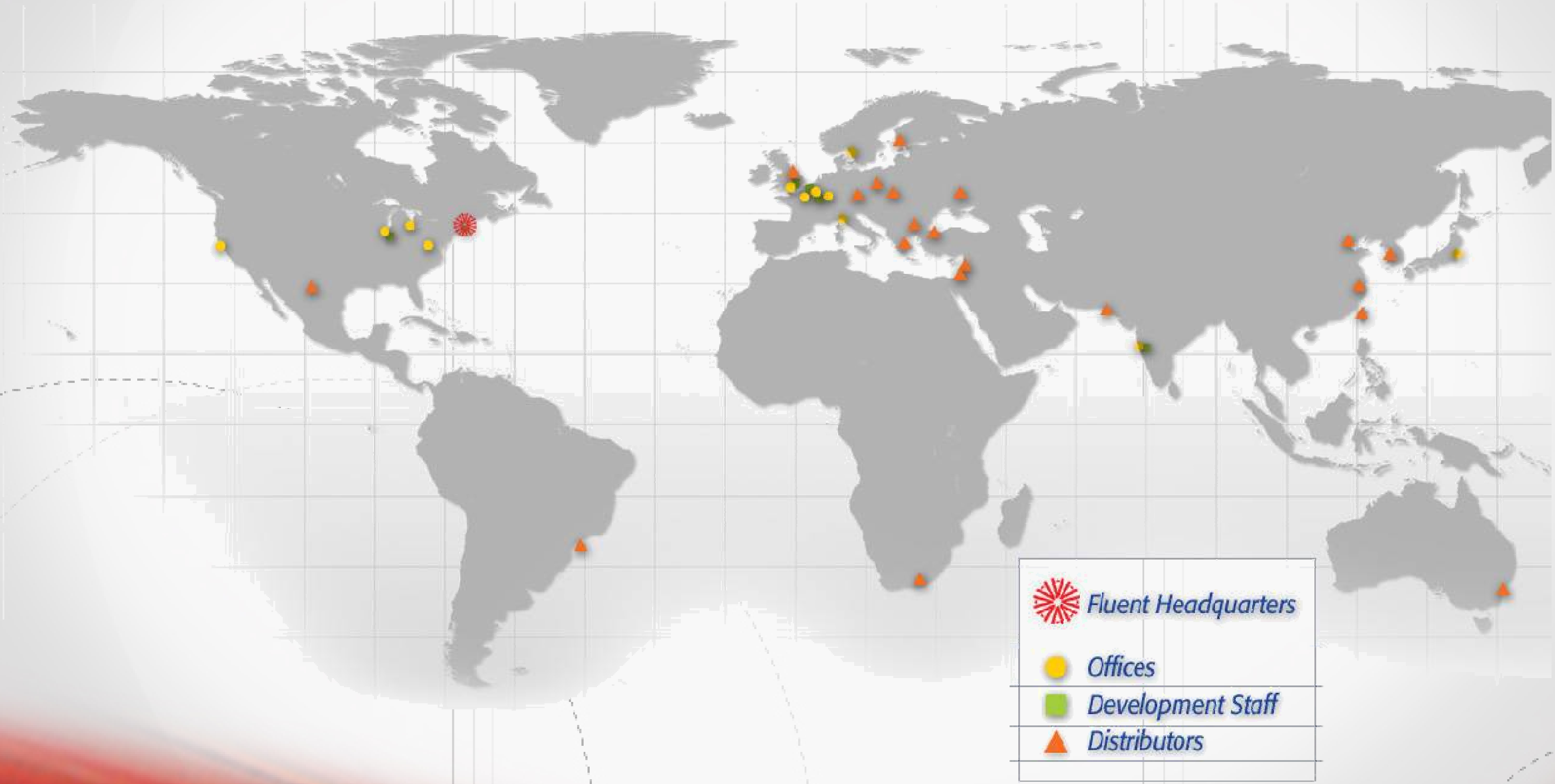
- 1988 Fluent group at Creare becomes Fluent Inc. - a separate company under the leadership of Bart Patel, the present Chief Executive Officer

Historical Background

- 1992 Fluent Germany opened and Fluent France formed
- 1999 Fluent Sweden opened

The Right Answer in CFD

Fluent – a Global Resource



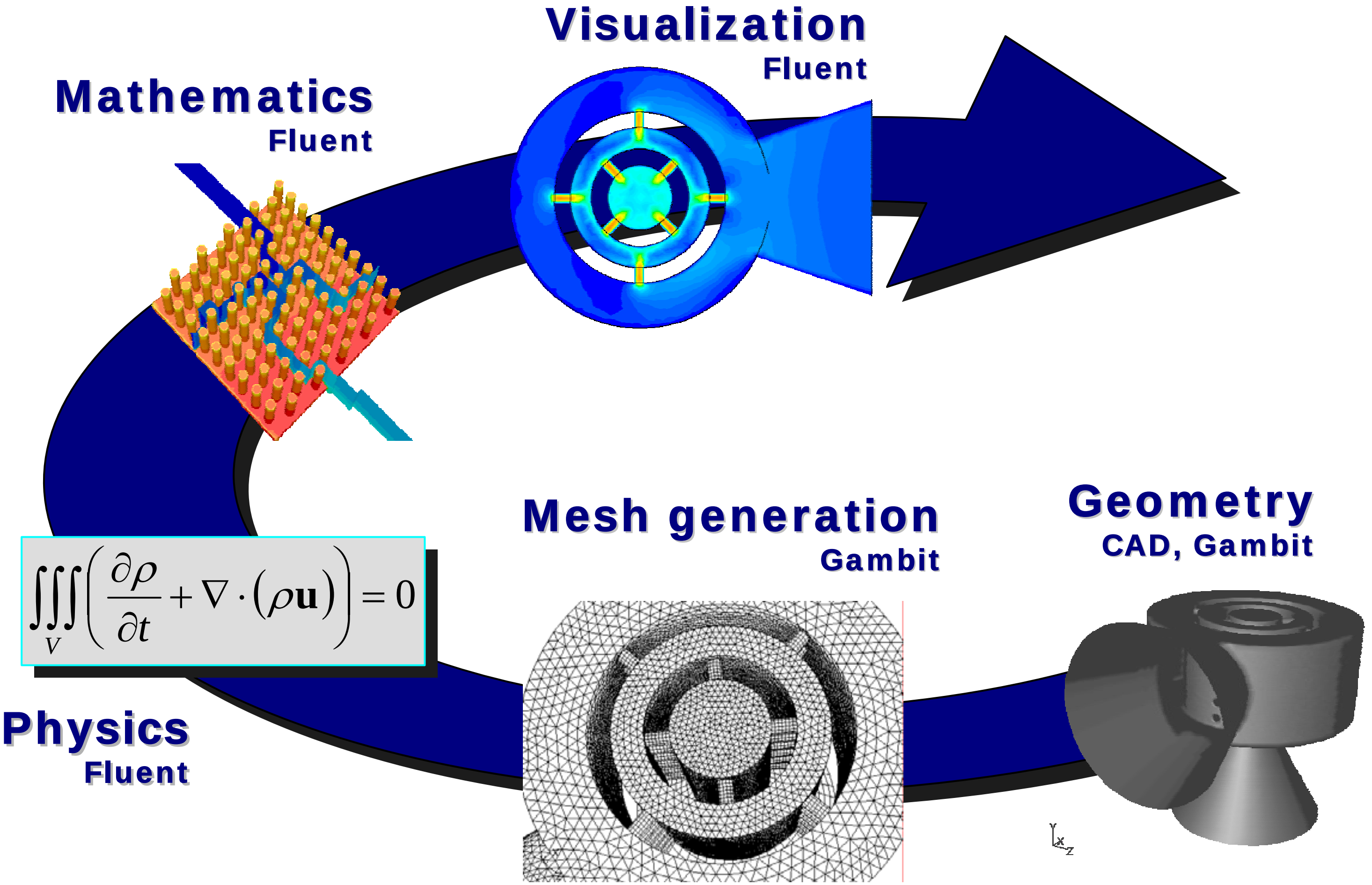
14 offices in 9 countries, almost 640 employees

FLUENT

The Right Answer in CFD

CFD Analysis

- Convection
- Conduction
- Turbulence
- Transient flow
- Combustion
- Radiation
- Rotation
- Multiphase
- Chemical reactions
- Phase change
- Acoustics
- Deforming mesh
- EMC
- Plasma flows
- ...



Visualization
Fluent

Mathematics
Fluent

Mesh generation
Gambit

Geometry
CAD, Gambit

Physics
Fluent

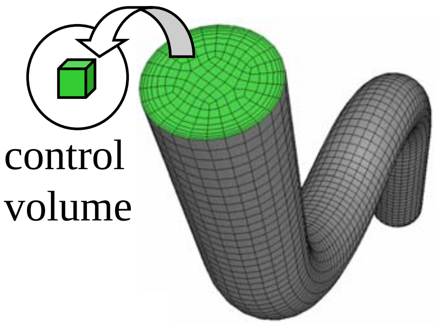
$$\iiint_V \left(\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) \right) = 0$$

What is CFD?

- Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, by solving numerically the set of governing mathematical equations.
  - Conservation of mass, momentum, energy, species, ...
- The results of CFD analyses are relevant in:
  - conceptual studies of new designs
  - Product development
  - troubleshooting
  - Redesign
- CFD analysis complements experimentation and theory.
  - Reduces the total effort required in the experiment design and data acquisition

CFD with Finite Volume Method?

- **Control volumes** or **Cells**.
- **General conservation** (transport) **equation** for mass, momentum, energy, etc.:

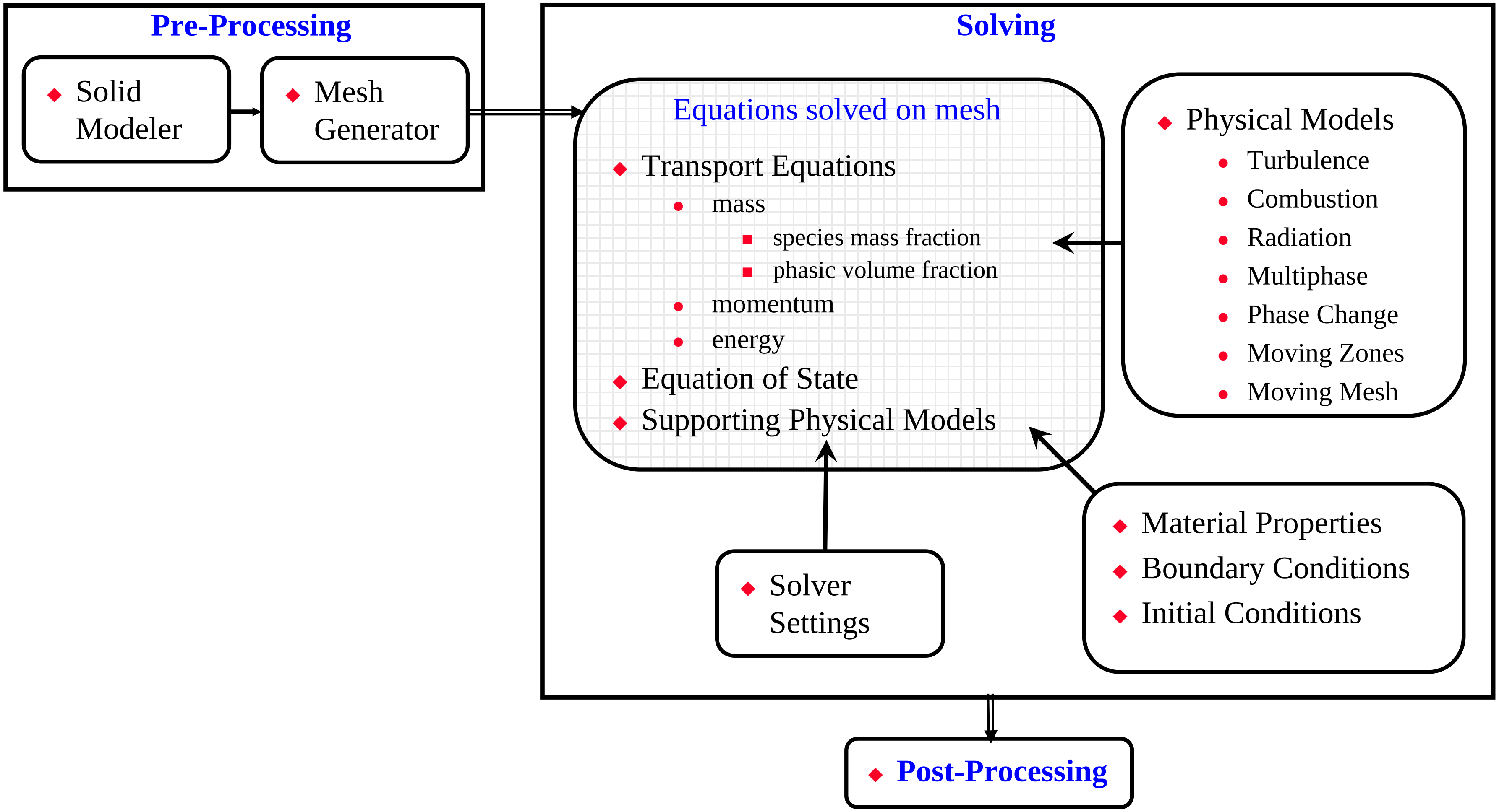


$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{generation}}$$

Eqn.	ϕ
continuity	**1**
x-mom.	**u**
y-mom.	**v**
energy	**h**

- **Partial differential** equations => **algebraic** equation system
- **Solved numerically**

The Tools



GAMBIT / Tgrid

Pre-Processing

- Solid Modeler
- Mesh Generator

Fluent

Solving

Equations solved on mesh

- Transport Equations
  - mass
    - species mass fraction
    - phasic volume fraction
  - momentum
  - energy
- Equation of State
- Supporting Physical Models

- Physical Models
  - Turbulence
  - Combustion
  - Radiation
  - Multiphase
  - Phase Change
  - Moving Zones
  - Moving Mesh

- Solver Settings

- Material Properties
- Boundary Conditions
- Initial Conditions

- **Post-Processing**

CFD Analysis: Basic Steps

- **Problem Identification** and Pre-Processing
  1. Define your **modeling goals**.
  2. Identify the **domain** you will model.
  3. Design and create the **grid**.
- **Solver Execution**
  4. **Set up** the numerical model.
  5. **Compute** and monitor the solution.
- **Post-Processing**
  6. **Examine** the results.
  7. Consider **revisions** to the model.

Define Your Modeling Goals

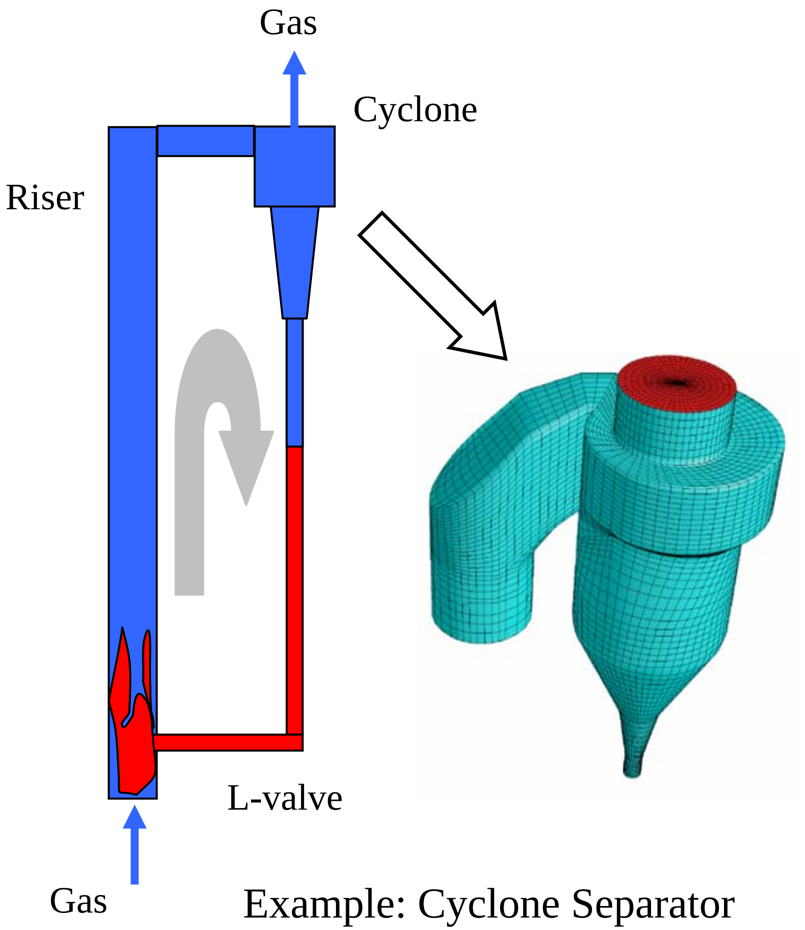
- Problem Identification and Pre-Processing
  1. Define your modeling goals.
  2. Identify the domain you will model.
  3. Design and create the grid.

- What results are you looking for, and how will they be used?
  - What are your modeling options?
    - What physical models will need to be included in your analysis?
    - What simplifying assumptions do you *have* to make?
    - What simplifying assumptions *can* you make?
- What degree of accuracy is required?
- How quickly do you need the results?

Identify the Domain

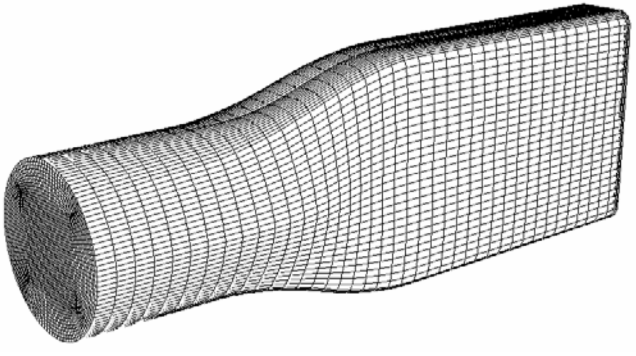
- Problem Identification and Pre-Processing
  1. Define your modeling goals.
  2. Identify the domain you will model.
  3. Design and create the grid

- Isolate a piece of the complete physical system
- Computational domain.
  - Boundary condition information
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?
- Is a 2D or axisymmetric approximation feaseable?

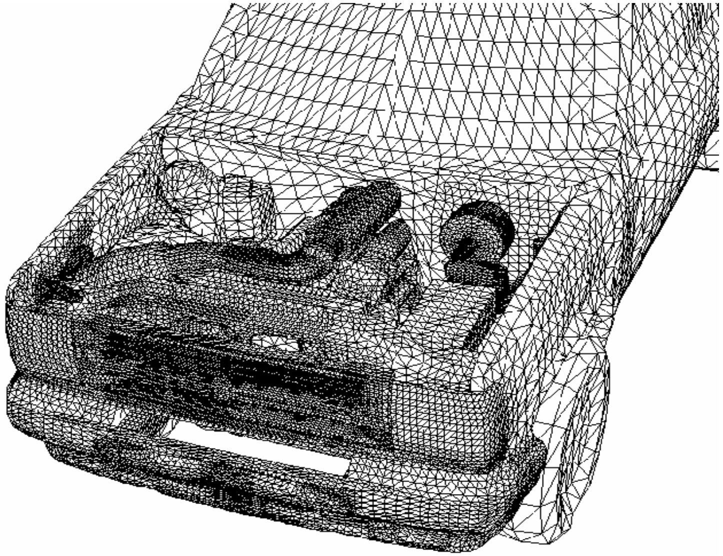


Example: Cyclone Separator

Tri/Tet vs. Quad/Hex Meshes



- For **simple** geometries, quad/hex meshes can provide higher-quality solutions with fewer cells than a comparable tri/tet mesh.
  - Align the gridlines with the flow.

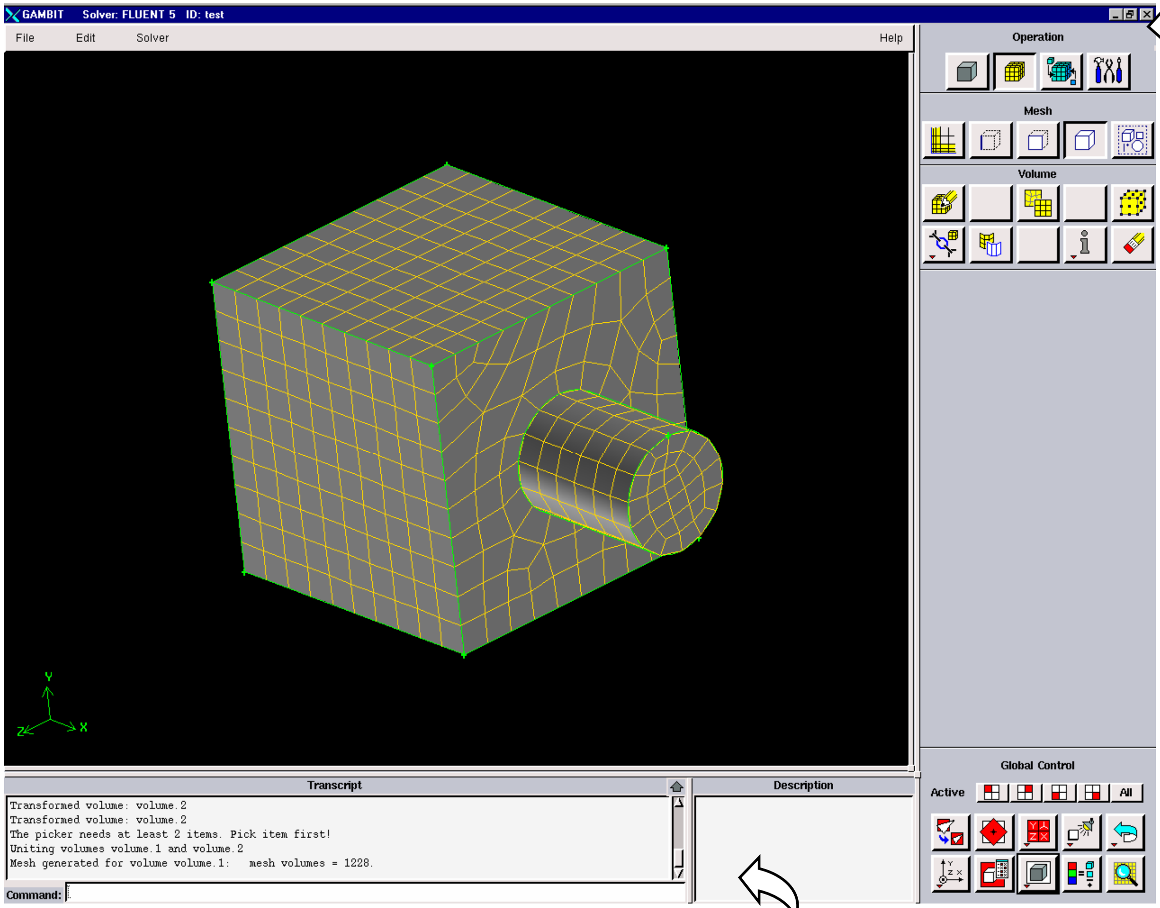


- For **complex** geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh.

Graphical User Interface (GUI)

Main Menu bar

Operation toolpad



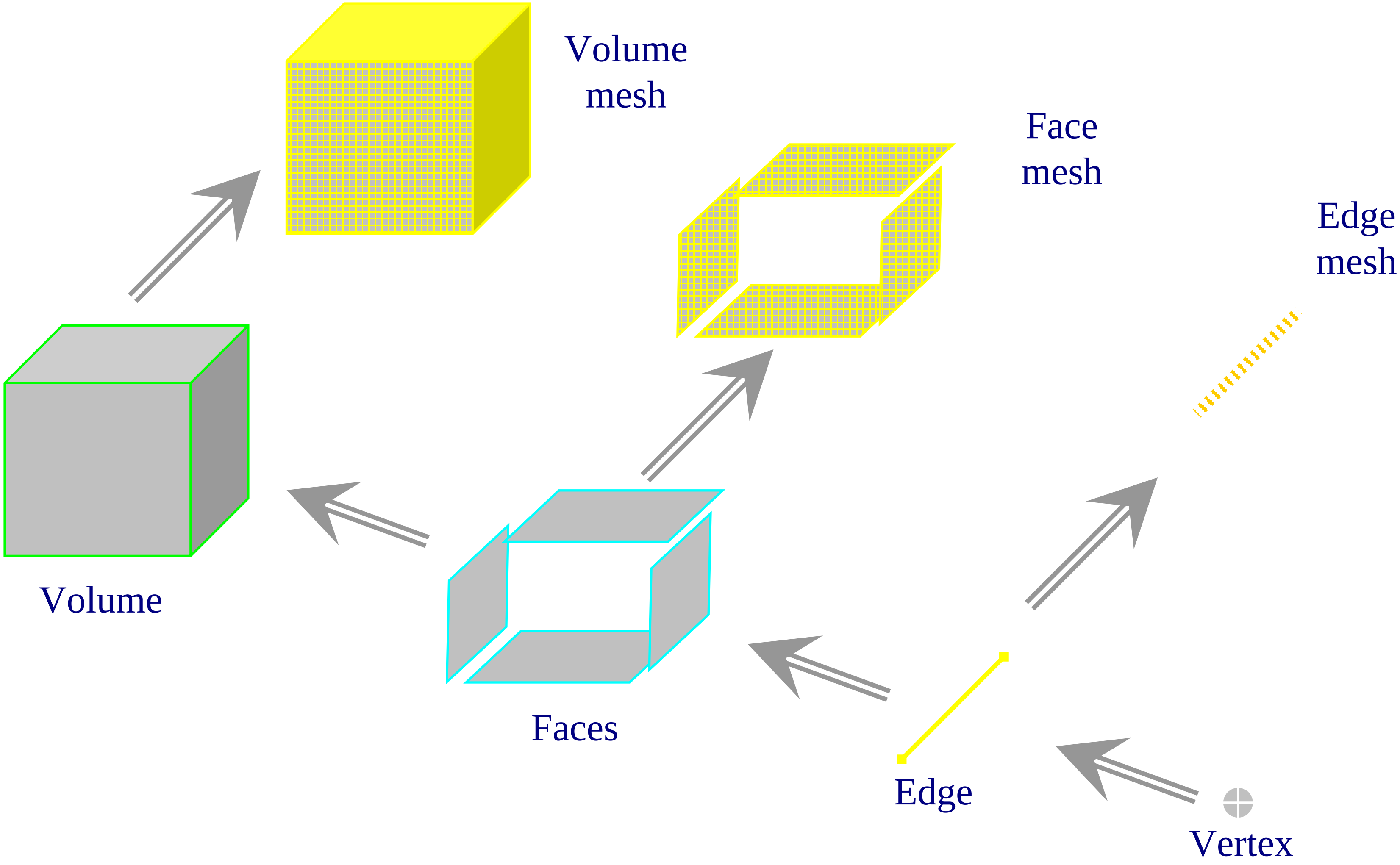
```
Transformed volume: volume.2
Transformed volume: volume.2
The picker needs at least 2 items. Pick item first!
Uniting volumes volume.1 and volume.2
Mesh generated for volume volume.1:   mesh volumes = 1228.
```

Command line

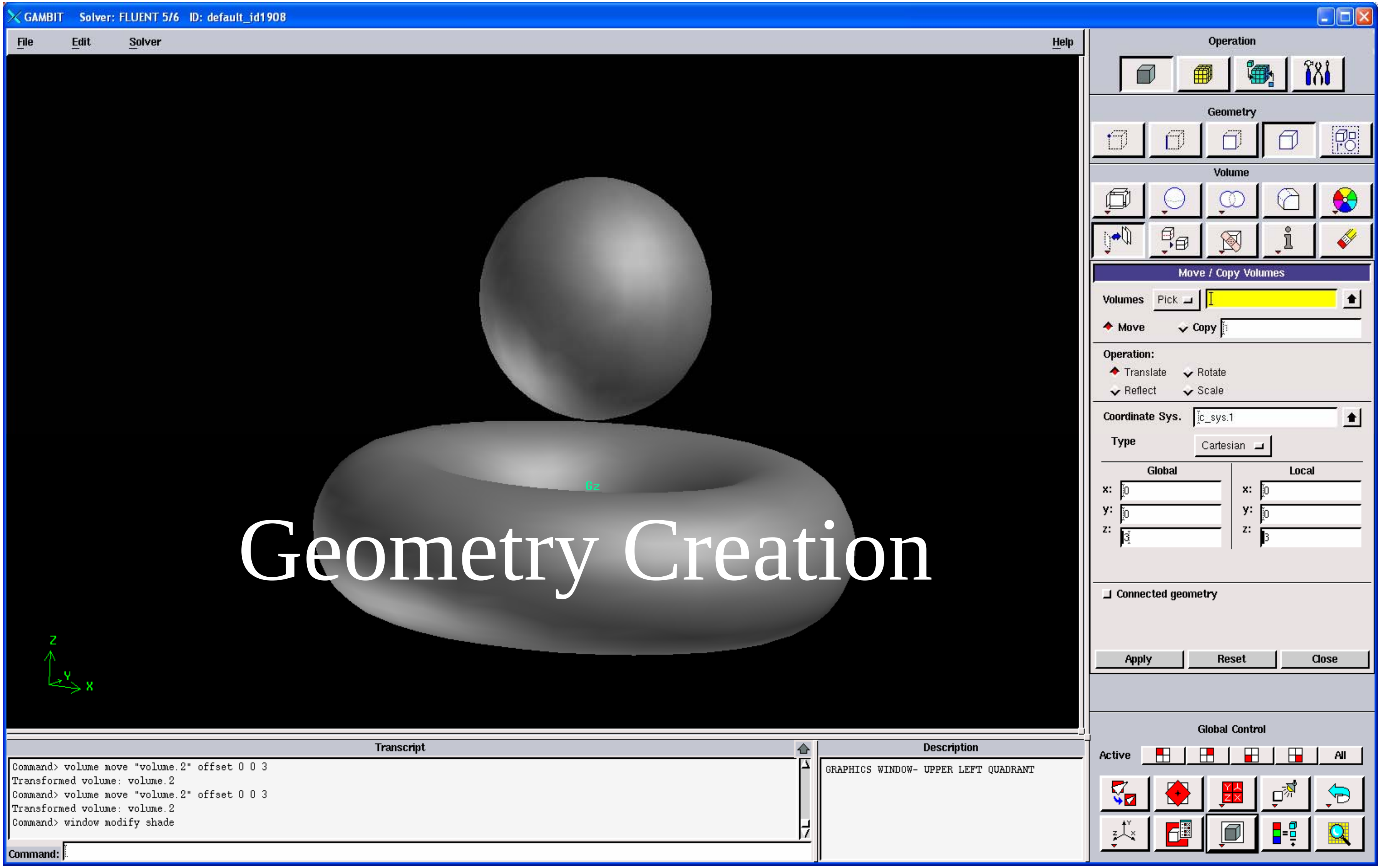
Description window

Global Control

Topology in Gambit



Geometry Creation



Volumes: Pick
◆ Move ✓ Copy

Operation:
◆ Translate ✓ Rotate
✓ Reflect ✓ Scale

Coordinate Sys.: c_sys.1
Type: Cartesian

	Global	Local
x:	0	0
y:	0	0
z:	3	3

☐ Connected geometry

```
Command> volume move "volume.2" offset 0 0 3
Transformed volume: volume.2
Command> volume move "volume.2" offset 0 0 3
Transformed volume: volume.2
Command> window modify shade
```

```
GRAPHICS WINDOW- UPPER LEFT QUADRANT
```

Geometry Creation

- ACIS - geometry engine ("kernel")
  - Provides tools for **"bottom-up"** creation by:

Vertex:	Add, Grid Snap, etc.
Edge:	Line, Arc, Ellipse, Fillet, B-spline, etc.
Face:	Wire Frame, Sweep, Net, etc.
Volume:	Wire Frame, Sweep, Face Stitch, etc.

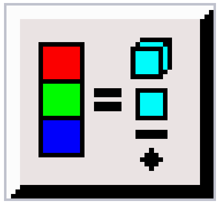
  - Provides tools for **"top-down"** creation by

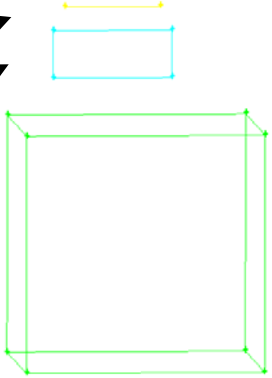
Face Primitives:	Rectangle, Circle, Ellipse
Volume Primitives:	Brick, Cylinder, Sphere, etc.
Volume/Face Booleans:	Unite, Subtract, Intersect
Volume/Face Decompose:	Split

  - Geometry creation **typically** involves use of **all tools**.

Geometry Creation (3)

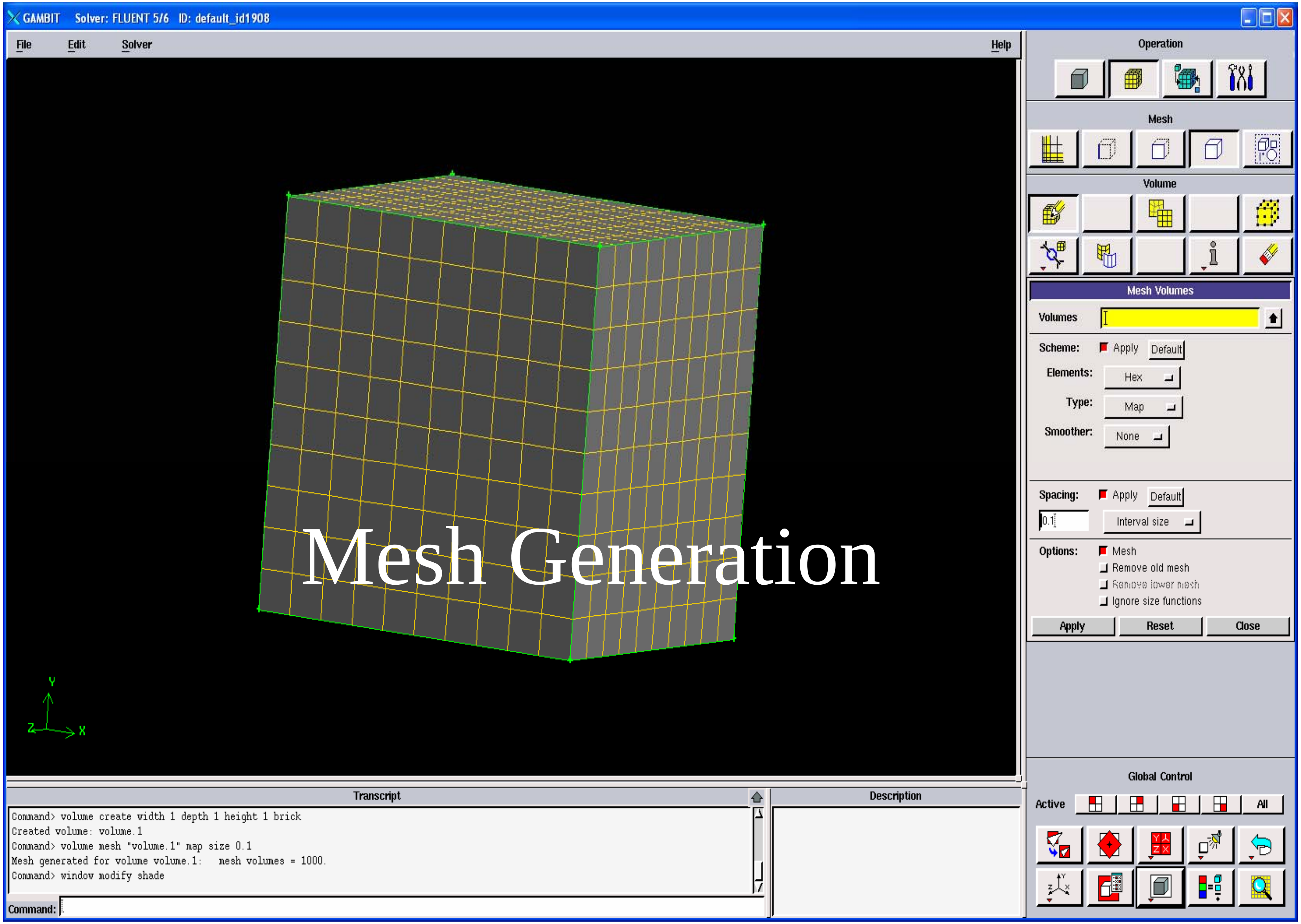
- **Color Identification**
  - Connected Vertices and Edges are colored according to the highest order entity to which they are connected. To.
  - The coloring scheme is:
    - Vertex (white)
    - Edge (**yellow**)
    - Face (**light blue**)
    - Volume (**green**)





- **Undo/Redo**:
  - **10 levels** of undo by default.
    - Undoes **geometry**, **meshing**, and **zoning** commands.
    - Description window provides command to be undone when mouse is passed over undo button.
  - **Left click** to execute visible button operation.
  - **Right click** to access options.

Mesh Generation

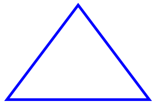


Volumes:
Scheme: ☒ Apply Default
Elements: Hex
Type: Map
Smoother: None
Spacing: ☒ Apply Default
0.1 Interval size
Options: ☒ Mesh
☐ Remove old mesh
☐ Remove lower mesh
☐ Ignore size functions

```
Command> volume create width 1 depth 1 height 1 brick
Created volume: volume.1
Command> volume mesh "volume.1" map size 0.1
Mesh generated for volume volume.1:   mesh volumes = 1000.
Command> window modify shade
```

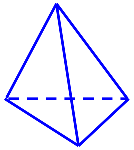
Command:

Design and Create the **Grid**

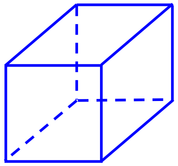


triangle

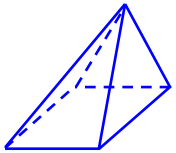
quadrilateral



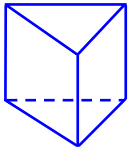
tetrahedron



hexahedron



pyramid

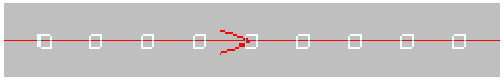


prism/wedge

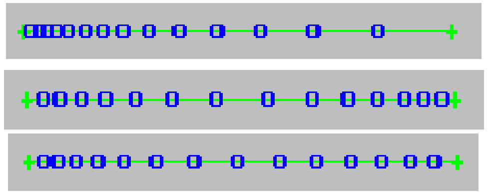
- **Quad/hex or tri/tet?**
- **Grid resolution?**
- **Sufficient computer memory?**

Edge Meshing

- Edge meshing form
  - **Picking**



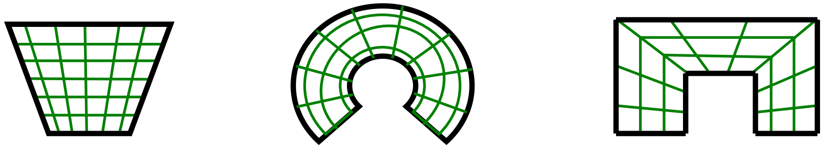
  - **Grading**
    - Single-sided grading
    - Symmetric grading
    - Asymmetric grading



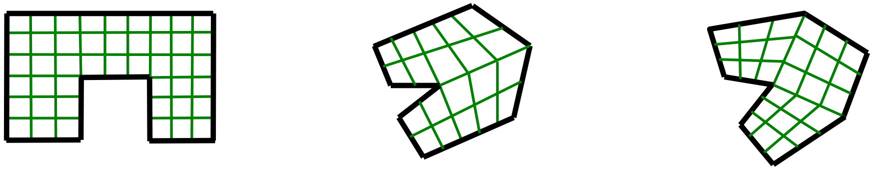
  - **Spacing**
    - Interval Count
    - % of edge length
    - Interval Sice
  - **Special characteristics**
  - **Options**

Face Meshing - Quad Examples

- Quad: Map

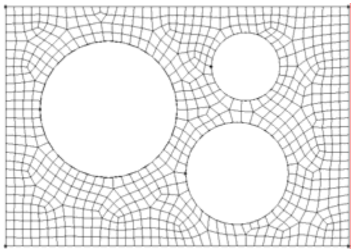


- Quad: Submap



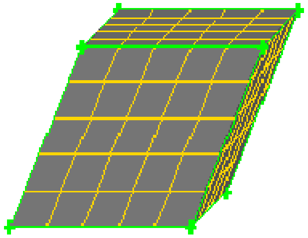
- Quad: Tri-Primitive

- Quad: Pave

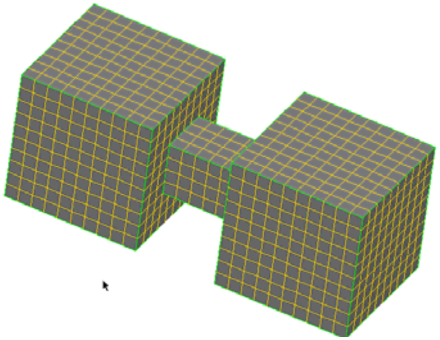


Volume Meshes - Hex Examples

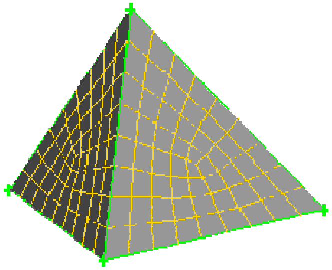
- Hex: Map



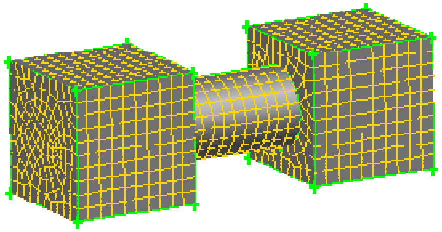
- Hex: Submap



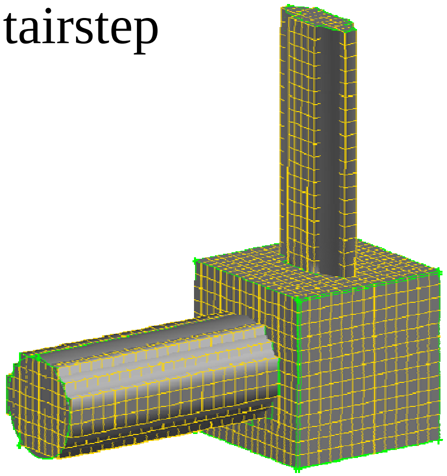
- Hex: Tet-Primitive



- Hex: Cooper



- Hex: Stairstep



Set Up the Numerical Model

- Solver Execution
  4. Set up the numerical model.
  5. Compute and monitor the solution.

Solving initially in 2D will provide valuable experience with the models and solver settings for your problem in a short amount of time.

- For a given problem, you will need to:
  - Select physical models.
  - Define material properties.
  - Prescribe operating conditions.
  - Prescribe boundary conditions.
  - Provide an initial solution.
  - Set up solver controls.
  - Set up convergence monitors.

Compute the Solution

- Solver Execution
  4. Set up the numerical model.
  5. Compute and monitor the solution.

- The discretized equations are solved *iteratively*.
- Convergence is reached when:
  - Changes in solution variables from one iteration to the next are negligible.
    - Residuals provide a mechanism to help monitor this trend.
  - Overall property conservation is achieved.
- The accuracy of a converged solution is dependent upon:
  - Accuracy of physical models
  - Grid resolution
  - Problem setup

A converged and grid-independent solution on a well-posed problem will provide useful engineering results!

Examine the Results

- Post-Processing
  6. Examine the results.
  7. Consider revisions to the model.

◆ Examine the results and extract useful data.

- Visualization Tools
  - Flow pattern?
  - Separation?
  - Shocks, shear layers, etc.?
  - Are key flow features being resolved?
- Numerical Reporting Tools
  - Forces
  - Heat transfer coefficients
  - Surface and Volume integrated quantities
  - Flux Balances

Examine results to ensure property conservation and correct physical behavior. High residuals may be attributable to only a few cells of poor quality.

Consider Revisions to the Model

- Post-Processing
  6. Examine the results.
  7. Consider revisions to the model.

- Are physical models appropriate?
  - Is flow turbulent?
  - Is flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- Are boundary conditions correct?
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- Is grid adequate?
  - Can grid be adapted to improve results?
  - Does solution change significantly with adaption, or is the solution grid independent?
  - Does boundary resolution need to be improved?