

# TFAWS 2005



---

## Techniques for Preparing Pro/Engineer Models for CFdesign Fluid Flow and Convective Heat Transfer Analysis

Caleb D. Fisher

NASA

John Glenn Research Center



# Outline

---



- I. Introduction
- II. Simplifying model geometry
- III. Creating fluid volumes
- IV. Utilizing Mechanical Volume Regions

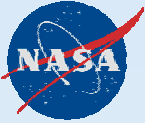


# Background

---



- ISS Avionics Box (Engineering Model)
    - Tasked with conducting thermal analysis
    - Needed more than “back of the envelope”
    - Pro/E model set up for design and manufacturing, not analysis
    - Model complexity surpassed that of previous CFdesign analyses
    - Result: Many lessons learned regarding simplifying complex models and constructing models w/ analysis in mind
    - Currently supporting redesign for flight
  - Techniques emphasize flow/convection
-

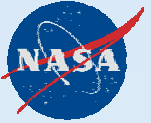


# Benefits of Coupling Pro/Engineer With CFdesign

---

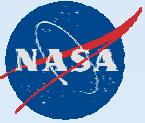


- Geometry not lost or skewed by approximation
  - Analysis utilizes the model by which the hardware will be constructed
- True associative relationship between design and analysis
  - Time and accuracy not sacrificed for rebuilding or updating analysis model
- Greater fidelity in preliminary design stage
  - Design engineer can do much better than “back of the envelope” without employing a CFD PhD



---

# Simplifying Model Geometry

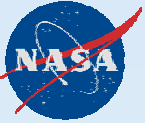


# Simplifying Model Geometry

---



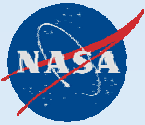
- Simplifying complex assemblies
  - Reduces unnecessary computational overhead
  - Promotes analysis convergence
  - Minimizes geometry/volume interferences
- Geometry analyzed by CFdesign needs only to be displayed in Pro/E environment
  - Avoids undesirable effects due to suppressing or deleting components not needed for analysis (even when “stepping”)



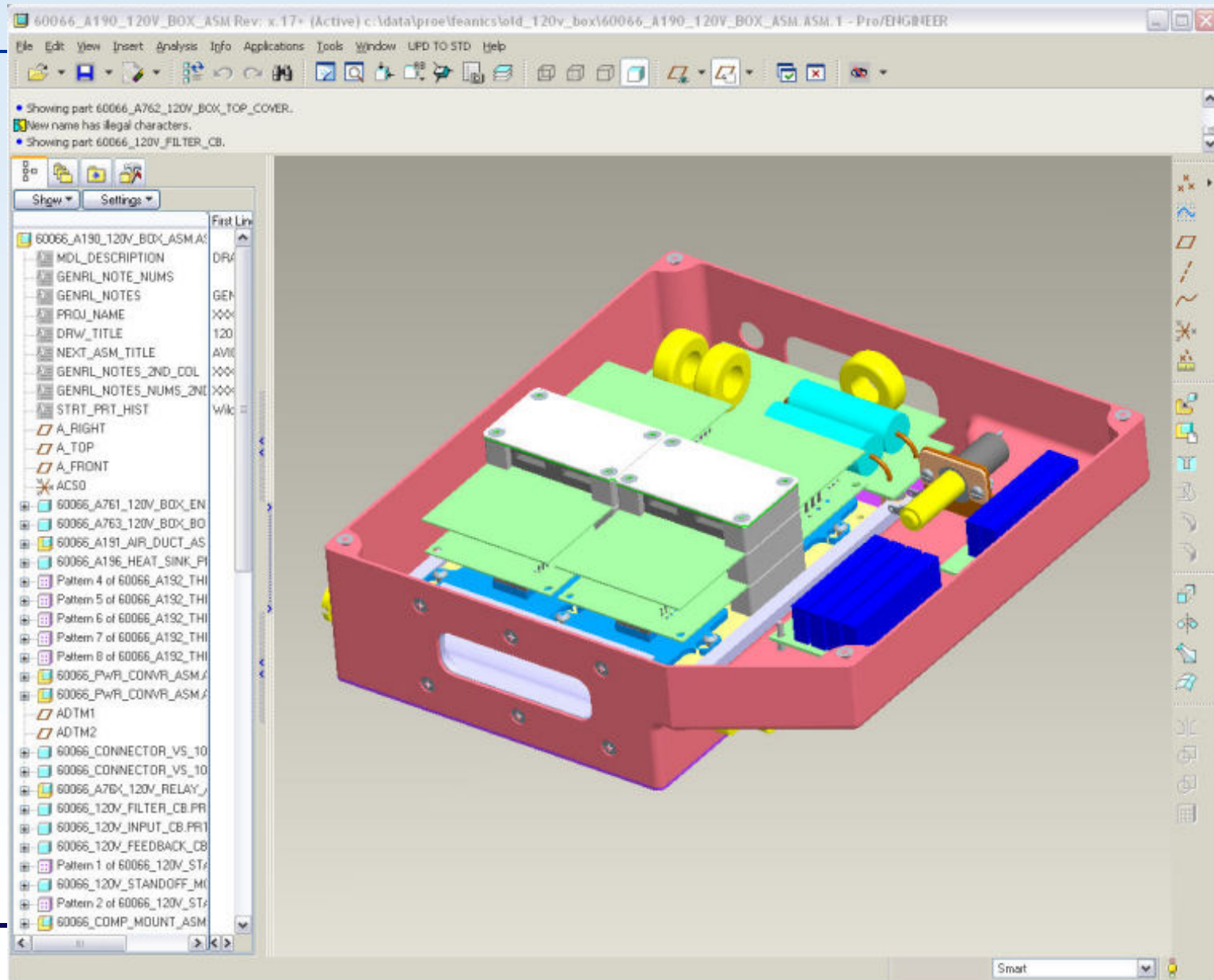
# Simplifying Model Geometry



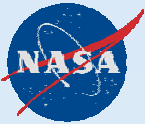
- Using ‘Simplified Reps’ in Pro/Engineer
  - Selected components are contained in single group which can be quickly hidden/redrawn
  - Reverting to previous configuration is 100% accurate
  - Multiple Simplified Reps are easily created and managed
  - View-based; no effect on actual model geometry
- Simplified Rep Creation (Wildfire 2.0)
  1. Select components to be removed from view
  2. Pick “View-Representation-Exclude”
  3. Run the View Manager, pick “Simp Rep”, and enter a name for new rep



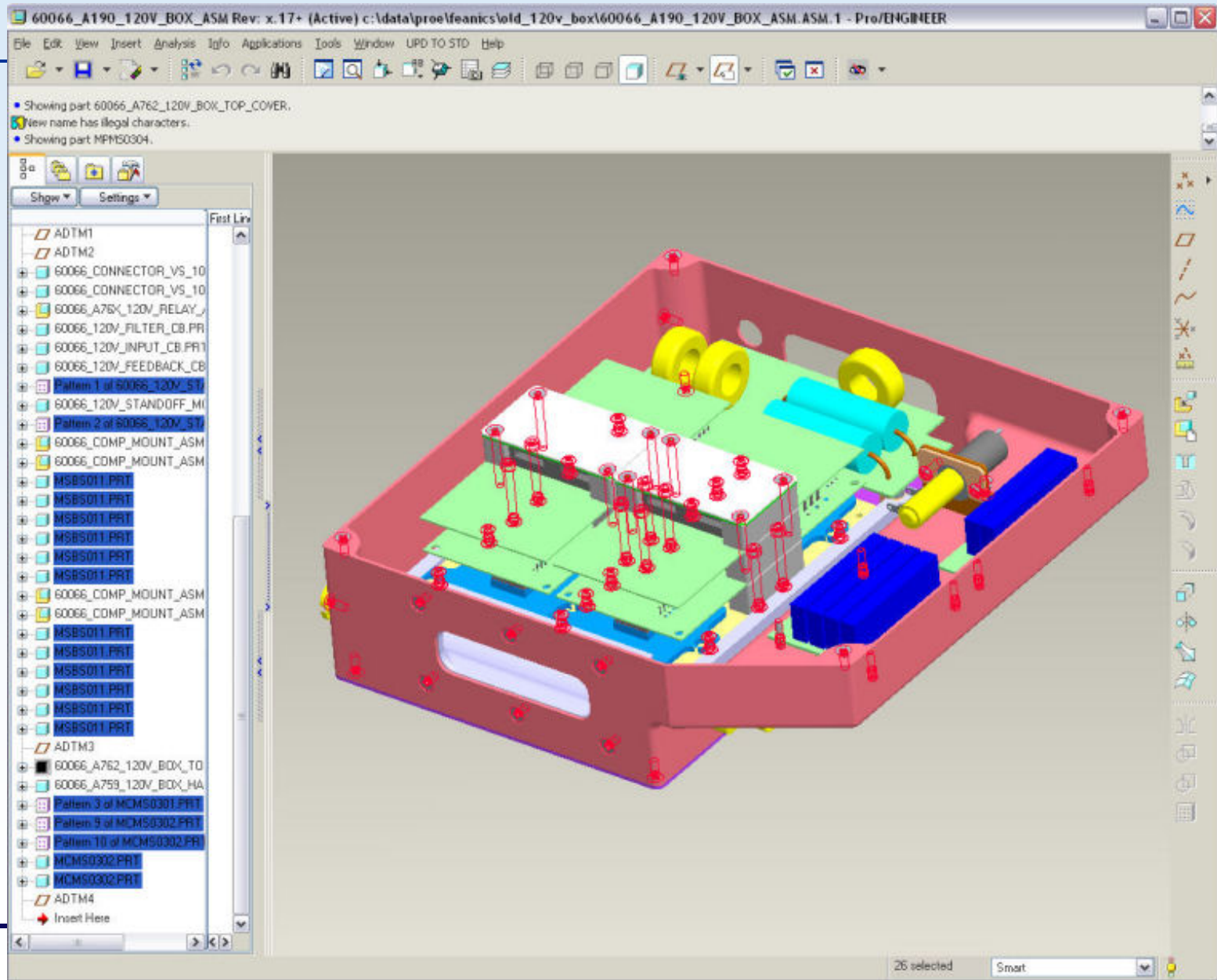
# Creating a Simplified Rep

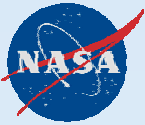




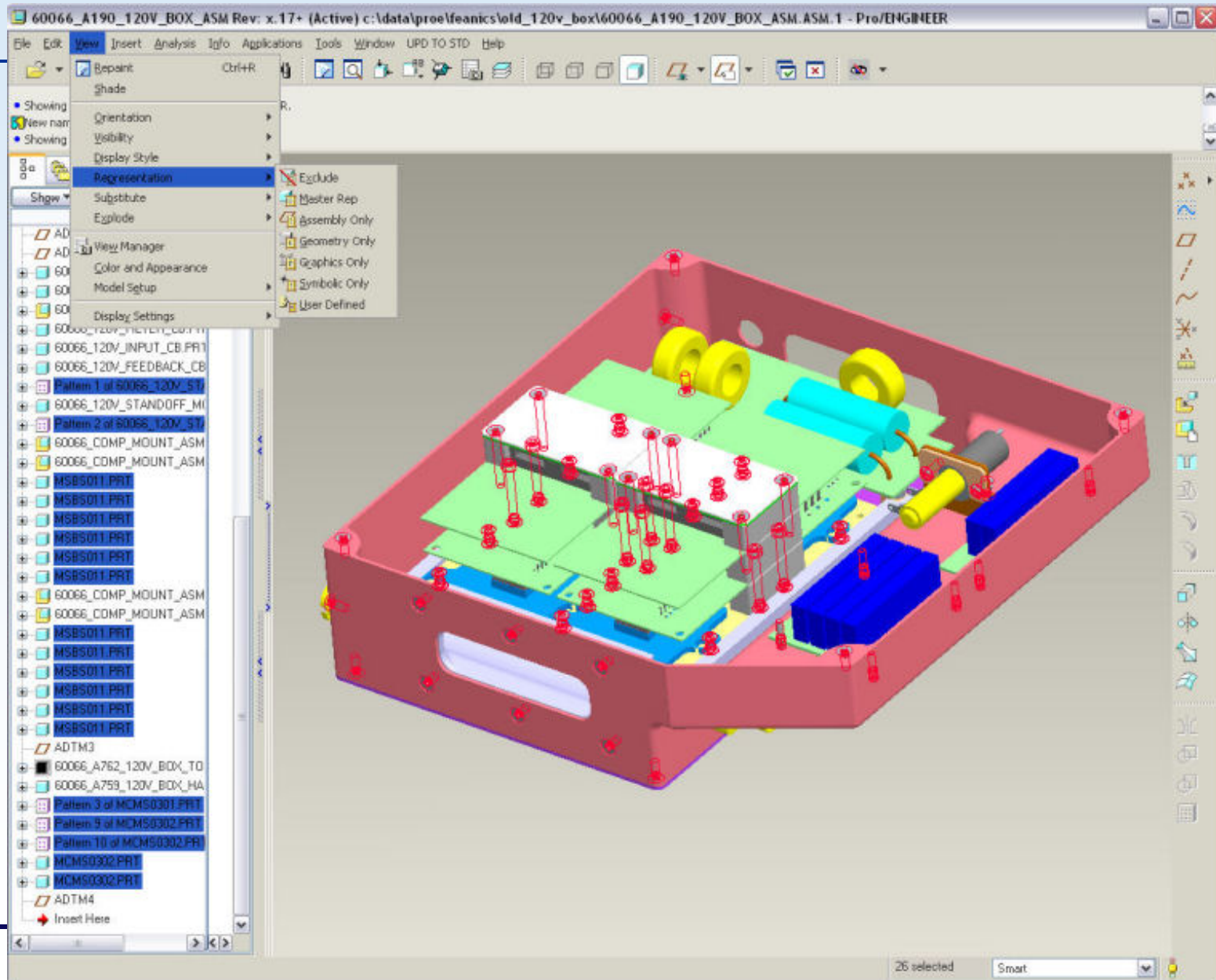


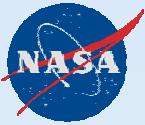
# 1. Select components to be removed from view



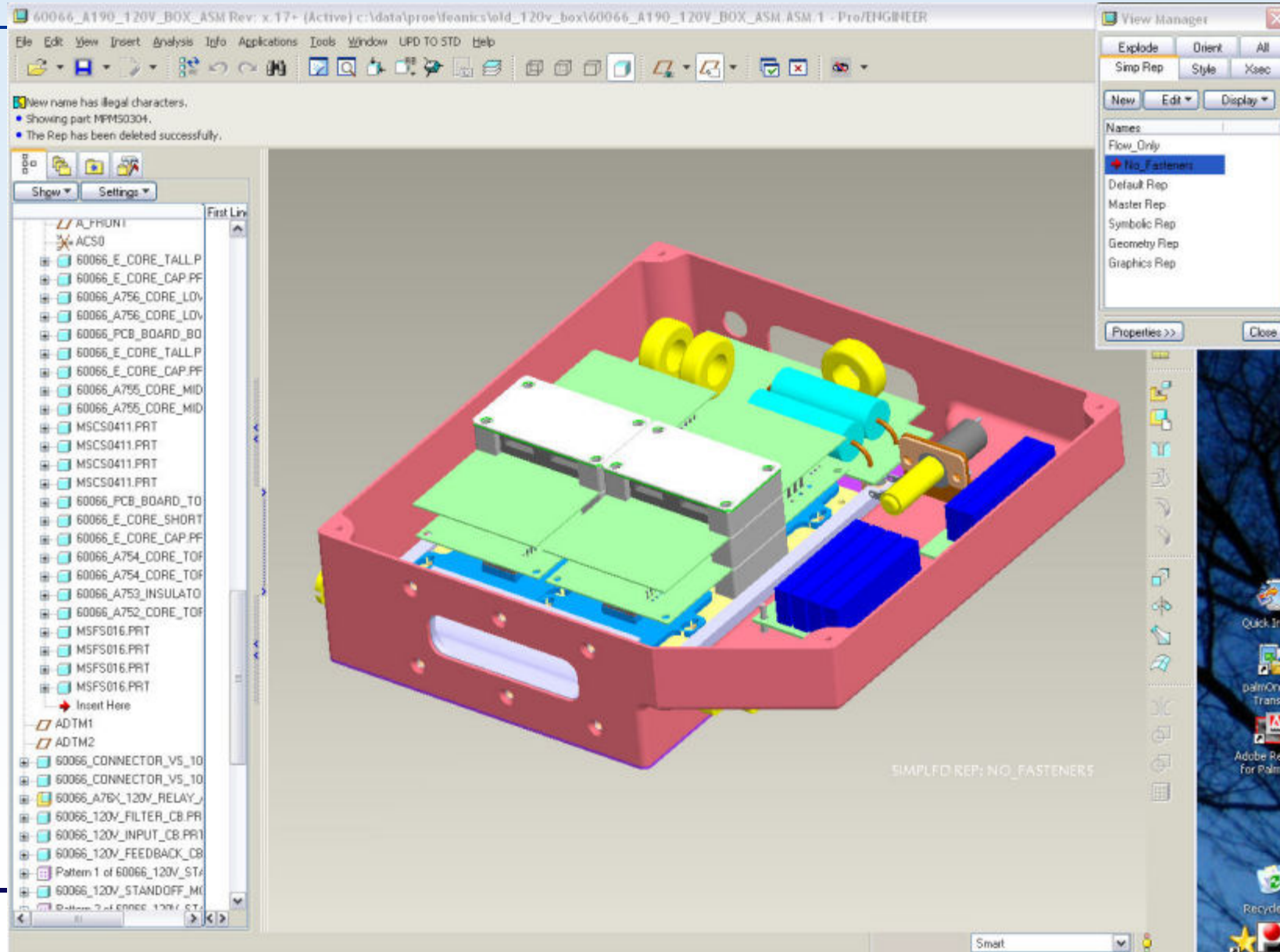


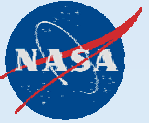
## 2. Pick "View-Representation-Exclude"





### 3. Run the View Manager, pick "Simp Rep", and enter a name





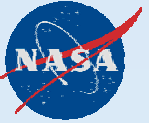
# Simplifying Model Geometry

---



## Reiterating benefits of 'Simplified Reps':

- Selected components are contained in single group which can be quickly hidden/redrawn
- Reverting to previous configuration is 100% accurate
- Multiple Simplified Reps are easily created and managed
- View-based; no effect on actual model geometry



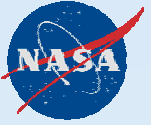
# Simplifying Model Geometry

---



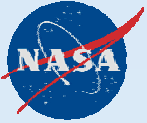
## Checking for remaining geometric interferences:

1. "Analysis - Model Analysis"
2. "Type" = Global Interference
3. Leave defaults → "Compute"
4. "Info" will produce full text
5. Exclude components from simplified rep on case-by-case basis (ugh)



---

# Creating Fluid Volumes



# Creating Fluid Volumes

---



- Automatic (“void filling”)
  - Preferred method
  - Advantages: Simple, fast, associative
  - Disadvantages: can be problematic w/ complex assemblies and/or flow paths
- Manual
  - Limited utility
  - Advantages: Full control over fluid geometry; can import fluid volume only
  - Disadvantages: No associativity, time intensive, human factors, process not trivial!



# Creating Fluid Volumes: Automatic

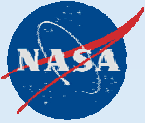
---



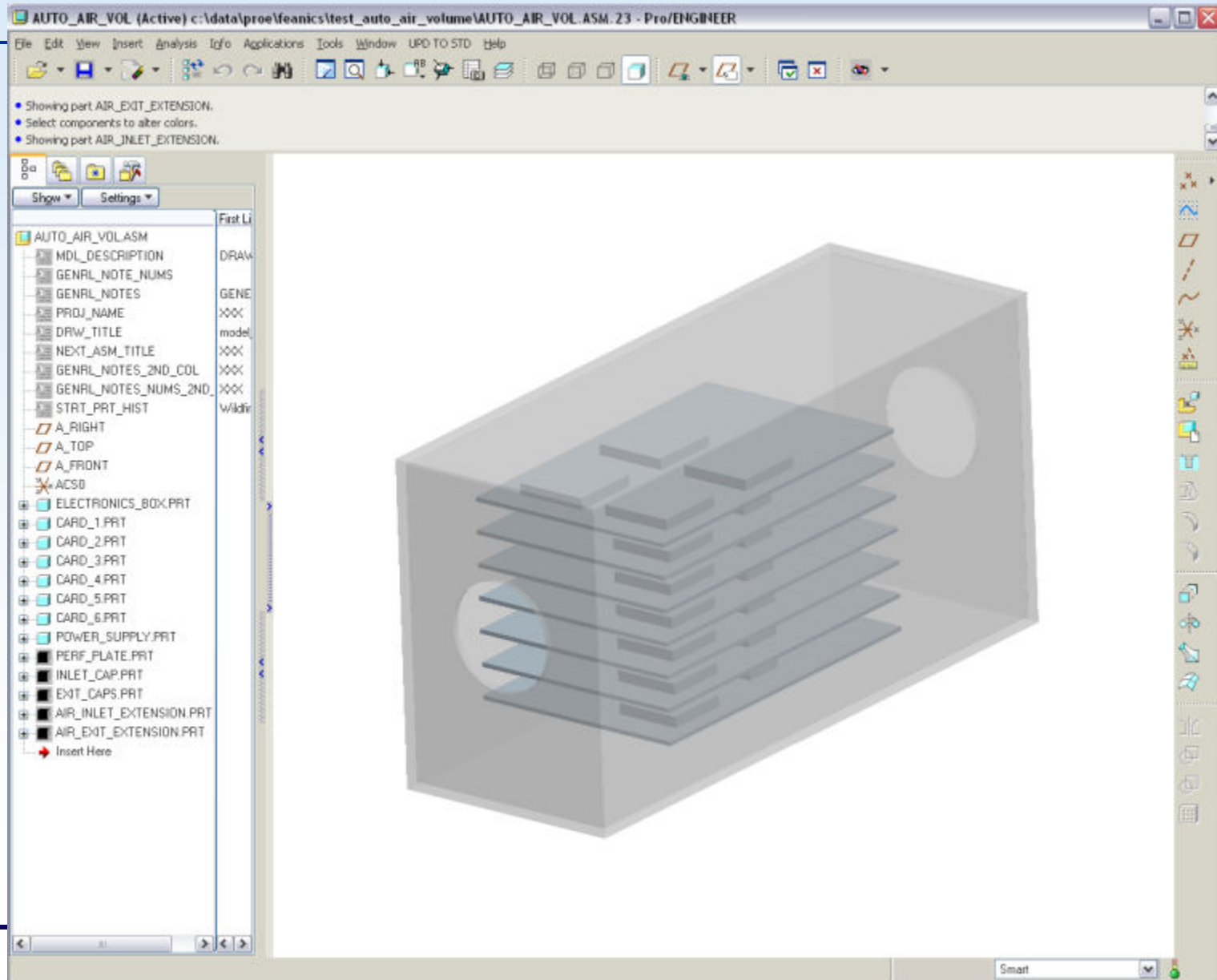
## Process:

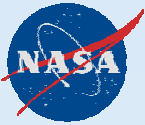
1. Create “caps” for all fluid inlets and exits
  - a) No internal components can protrude into caps
  - b) Caps must be flush with existing solid geometry
2. Create flow extensions for inlets and exits
  - a) Promotes analysis convergence (CFdesign assumes fully-developed flow)
  - b) Exit is mandatory; inlet is optional
  - c) Extrude directly from caps when practical
3. Create ‘Fluid Analysis’ component layer and transfer recently-created components
  - a) Isolates solid geometry not part of actual hardware



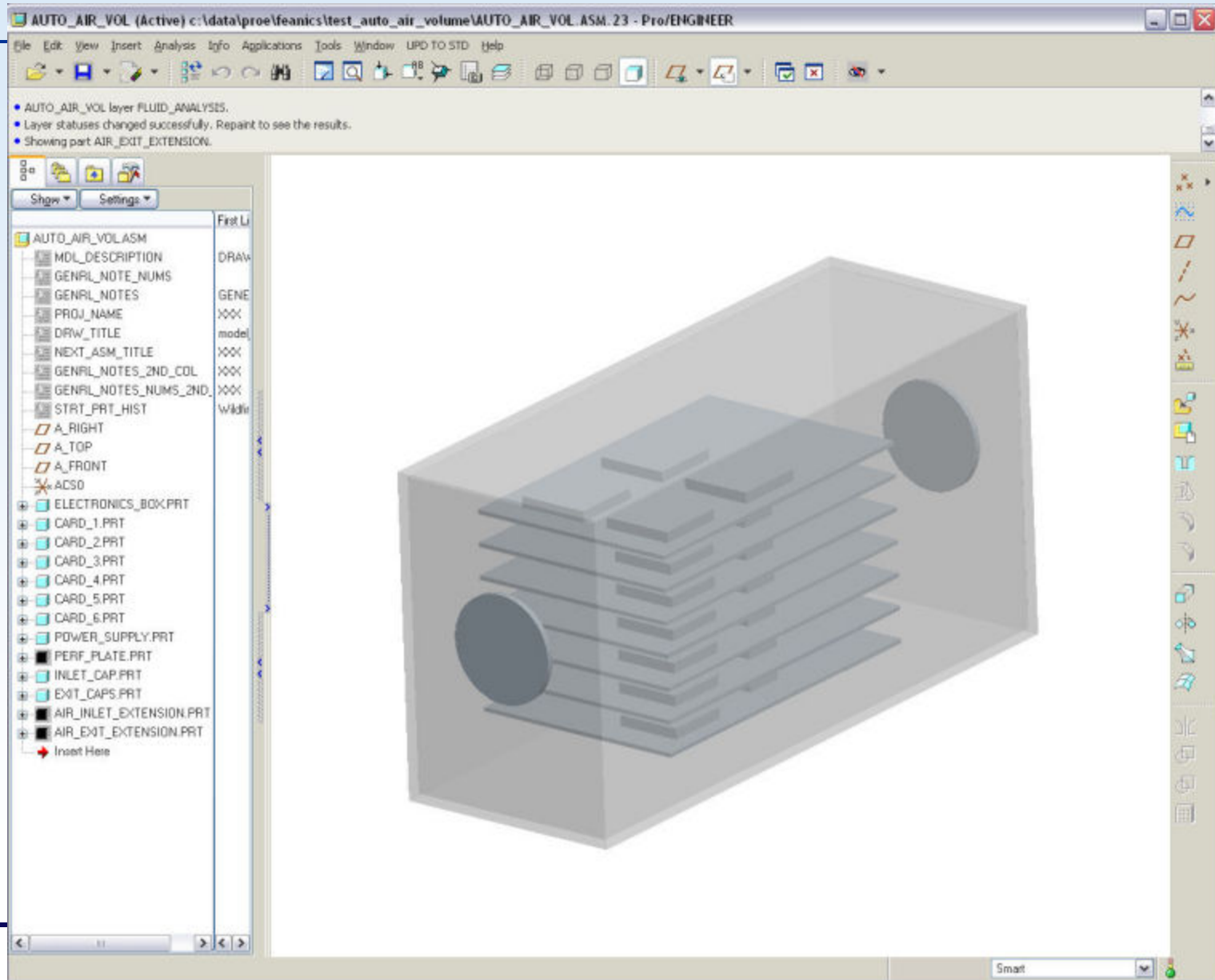


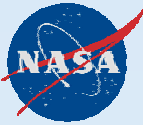
# Generic Electronics Box



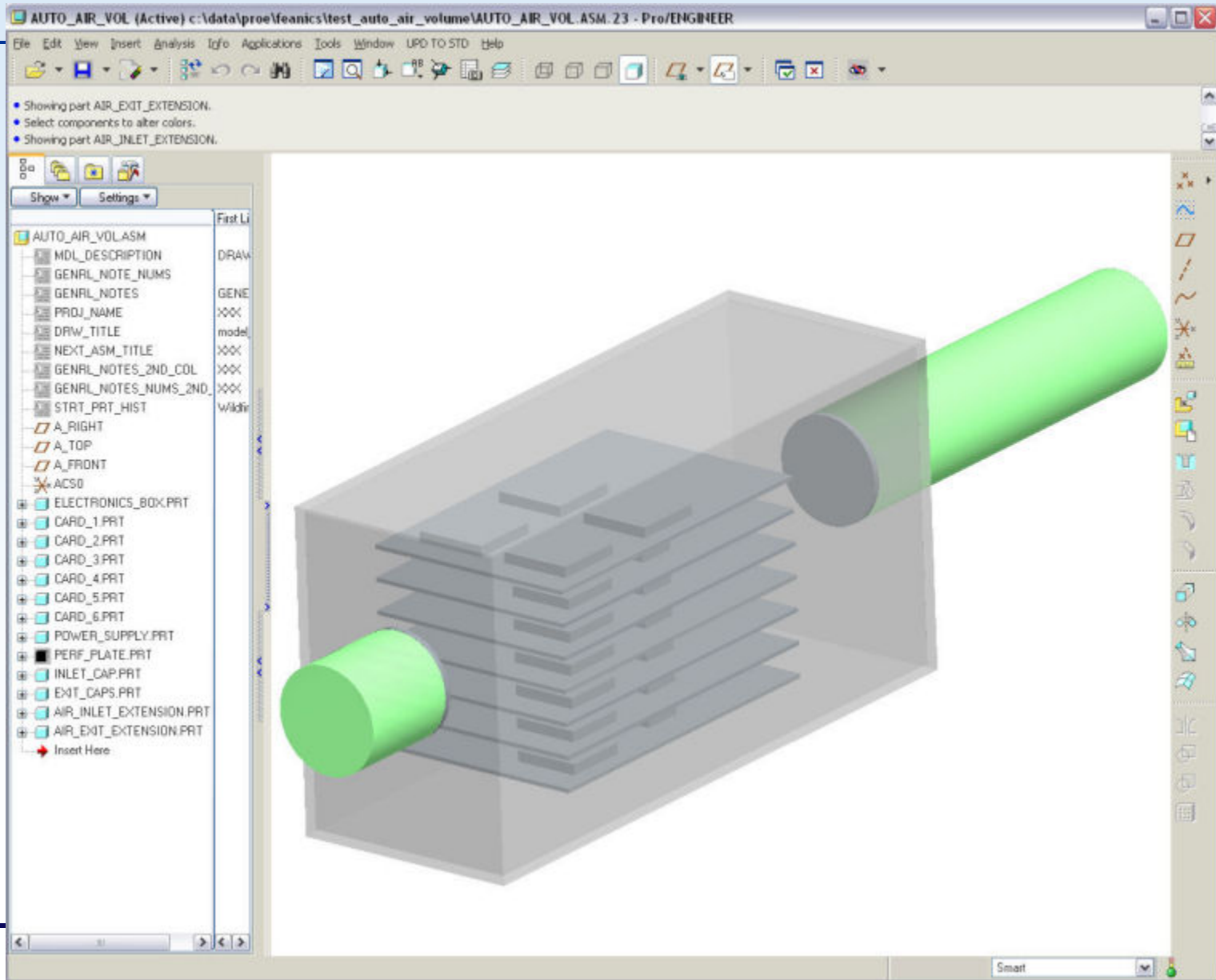


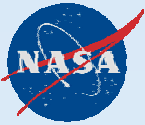
# 1. Create "caps" for all fluid inlets and exits





## 2. Create flow extensions for inlets and exits





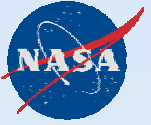
### 3. Create 'Fluid Analysis' component layer



The screenshot shows the SolidWorks CAD environment. The main window displays a 3D model of a duct assembly with two green cylindrical inlet/outlet ports. The left-hand side shows the Feature Tree with the 'FLUID\_ANALYSIS' layer selected. The 'Layer Properties' dialog box is open on the right, showing the following table:

Item	Status
C14(INLET_CAP)	+
C15(EXT_CAPS)	+
C16(AIR_INLET_EXTENSION)	+
C17(AIR_EXIT_EXTENSION)	+

The dialog box also includes buttons for 'Include...', 'Exclude...', 'Remove', 'Info...', 'Pause', 'OK', and 'Cancel'.



# Creating Fluid Volumes: Automatic

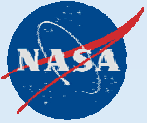
---



## 4. Ready for import into CFdesign

A couple tips:

- “PROTOCOL FILLVOIDS 1” must be set in `cfdesign_flags.txt`
- Be aware of effects on model properties, such as overall mass and BOM



# Creating Fluid Volumes: Manual

---

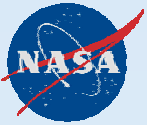


In a nutshell:

1. Create a “bulk” fluid volume
2. “Cut out” flow paths using existing components

Specifically:

1. Create simplified reps of all sub-assemblies
2. Create geometric “Shrinkwrap” of each simplified rep
3. Assemble shrinkwrapped components into a new assembly for making “cut-outs”
4. Create a bulk fluid volume (in above assembly)
5. Cut to final shape using “Adv.Util – Cut Out”

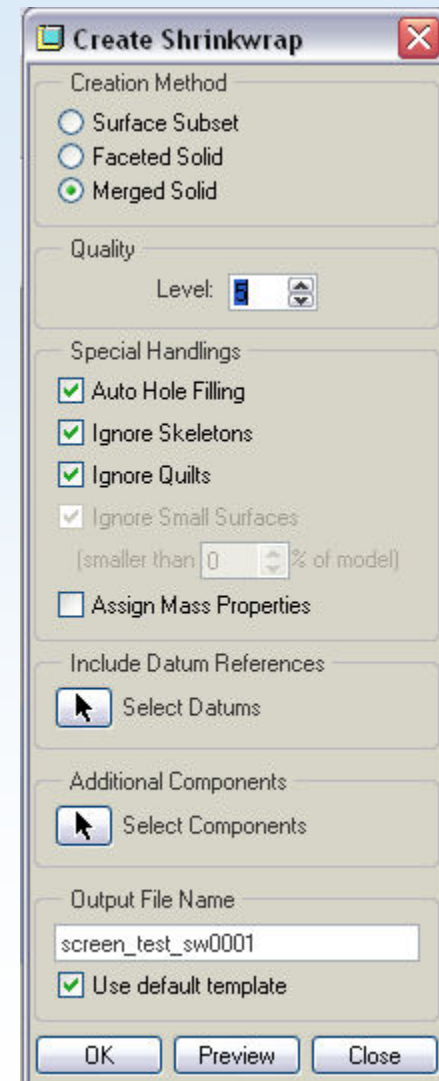


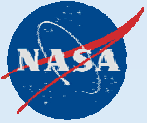
# Creating Fluid Volumes: Manual



## Creating geometric “Shrinkwraps” in Pro/E:

1. Open the sub-assembly and switch to the simplified rep created in Step 1
2. “Save A Copy” and select “Shrinkwrap” as Type
3. Select the “Merged Solid” option
4. “Quality”: Start w/ 5 and increase if necessary
5. Use “Auto Hole Fill” option with discretion
6. Select “Preview” and check # components selected (this tells you if Quality level is sufficient)
7. “Create”





# Creating Fluid Volumes: Manual

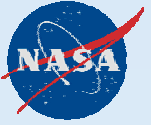
---



“Advanced Utility - Cut Out”: Removing material from one component using the geometry of another

1. Under the “Edit” menu, select “Component Operations”
2. Choose “Cut Out” from the menu that appears
3. First select the part *to be cut* (fluid volume)
4. Then select the cutting part (shrinkwrap component)
5. “Done”





---

# Utilizing Mechanical Volume Regions



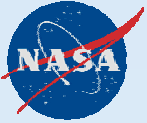
# Utilizing Mechanical Volume Regions



---

## Uses for Volume Regions in CFdesign:

- Segregating complex volumes to allow for more discrete assignment of:
  - Boundary and initial conditions
  - Thermal, fluid, and other material properties
- Modeling and analyzing screens/meshes/flow resistances
  - Simplifies solid model, thus reducing overhead
  - Improves analysis accuracy and promotes analysis convergence
  - Allows efficient trade studies regarding FAR's using CFdesign's 'Design Center'
- Note: In order for CFdesign to recognize the volume regions, they can neither overlap nor share a surface



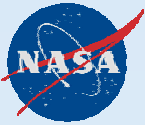
# Utilizing Mechanical Volume Regions

---

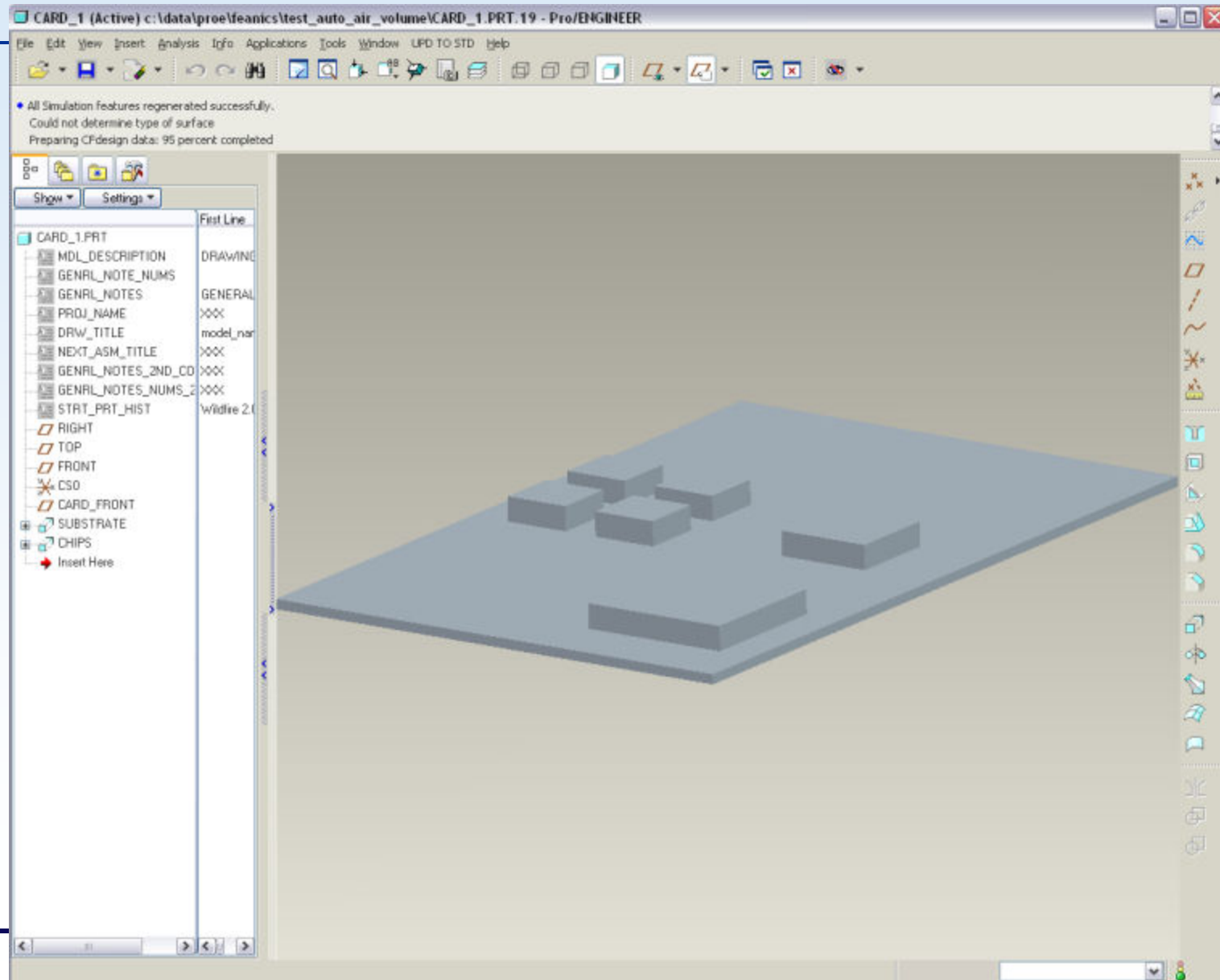


## Creating Volume Regions:

1. Start 'Mechanica' Application
2. "Insert – Volume Region"
3. Choose feature creation method (extrude, sweep, etc)
  - a) Note: Unfortunately you cannot use existing geometry
4. Select a surface to sketch upon and create geometry following normal Pro/E protocols
5. The new volume region shows up under Simulation Features and is recognized discretely when imported into CFdesign



# Volume Region Example: Circuit Board





# CFdesign - Before Volume Region



CFdesign 7.0 Analysis: erase

File Help

Feature Tree

- Analysis - erase
  - Length Units: meter
  - Coordinate System: Cartesian 3D
  - Parts
  - Materials
  - Boundary Conditions
  - Initial Conditions
  - Mesh Size
  - Groups

Task Dialogs

Boundary Initial

Model Entity Selection

Volume  Surface  Edge

Selection Basis: Direct

1-[None]

Boundary Condition

Type: Heat Generation

Units: W/m3

Steady State  Transient

Temperature Dependent

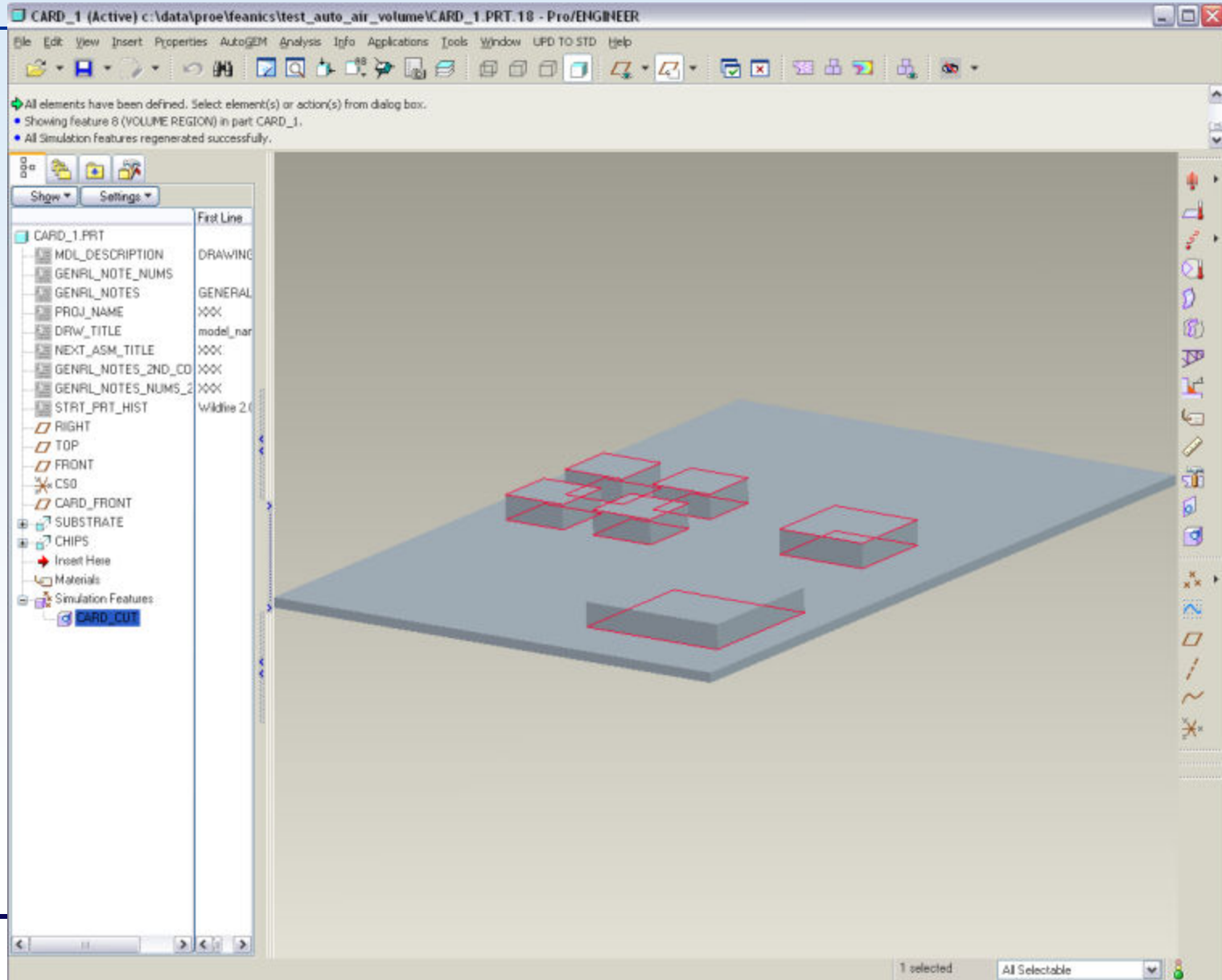
Heat Generation:

Apply Delete Delete All Help

Time: 0 Model



# Isolate Components With Volume Region





# CFdesign – After Volume Region



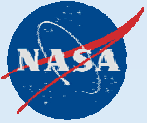
The screenshot displays the CFdesign 7.0 software interface. The main window is titled "CFdesign 7.0 Analysis: erase". The interface includes a menu bar (File, Help), a toolbar with various icons, and a Feature Tree on the left. The Feature Tree shows the following structure:

- Analysis - erase
  - Length Units: meter
  - Coordinate System: Cartesian 3D
  - Parts
  - Materials
  - Boundary Conditions
  - Initial Conditions
  - Mesh Size
  - Groups

The Task Dialogs panel is open to the "Boundary" tab, showing the "Initial" sub-tab. The "Model Entity Selection" section has "Volume" selected, and the "Selection Basis" is set to "Direct". The "Boundary Condition" section is configured as follows:

- Type: Heat Generation
- Units: W/m3
- Steady State (selected)
- Transient (unselected)
- Temperature Dependent (unchecked)
- Heat Generation: [Empty text box]

Buttons for "Apply", "Delete", "Delete All", and "Help" are visible at the bottom of the dialog. The 3D model in the center shows a red volume region with several grey rectangular blocks inside. A coordinate system is visible in the bottom right corner of the model area, with axes labeled X, Y, and Z. The text "Time: 0 Model" is displayed at the bottom left of the model area.



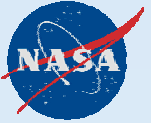
# Acknowledgements

---



- Bryan Fraser, NASA, Glenn Research Center
  - Puts the 'Engineer' in Pro/E
  - Instrumental in manual fluid volume creation method





---

Questions?