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# *Using STEP for Production Data Transfer*

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*NSRP Joint Panel Meeting*

*July 31, 2007*

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# Overview

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- *Defining STEP as a Universal Tool*
- *Processing Data for Shipbuilding*
- *Transferring Model Data*
- *Advantages of STEP*
- *A Critical Need for STEP*
- *Conclusions*

## *Defining STEP as a Universal Tool*

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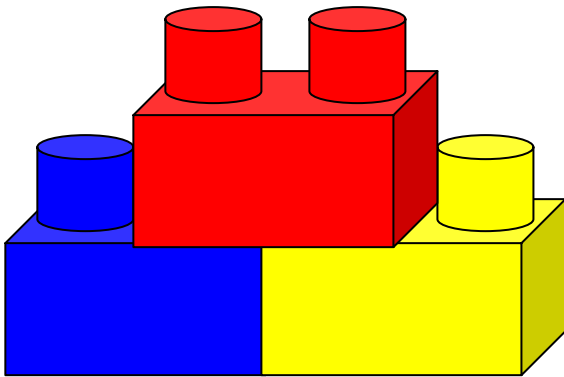
- *STandard for the Exchange of Product Model Data*
- *Started by the International Standards Organization*
- *Built on the success of previous neutral file types such as IGES, SET, and VDAFS*
- *Supported by most, if not all, major CAD software programs currently in use*

# *A 3D Transfer Standard with Customization*

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- *A 3D format*
- *A tool for transferring solid and surface geometry*
- *Contains model intelligence*
- *Different protocols hold different types of data*
  - *AP203 – General 3D Solid Exchange*
  - *AP214 - Core Data for Automotive Mechanical Design Processes*
  - *AP227 - Plant Spatial Configuration*

# Processing Data for Shipbuilding



- *Individual parts are organized into groups of assemblies*
- *Parts are designed on a primary CAD tool*
- *Different CAD programs provide different functionality*
- *STEP allows the transfer of 3D geometry to a variety of CAD software programs*

# Transferring Model Data

- STEP files are created from converting the proprietary design geometry into solids and surfaces

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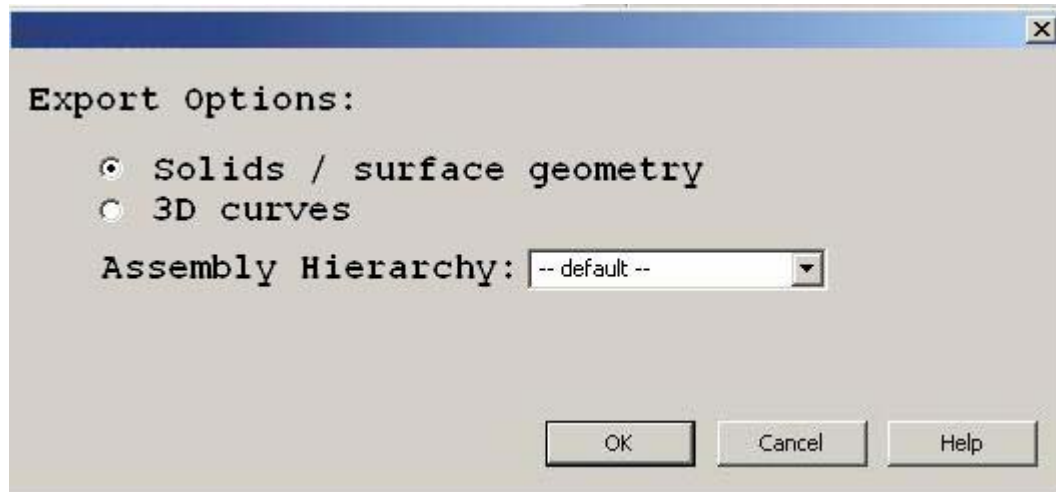
Proprietary CAD format

```
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HEADER;
FILE_DESCRIPTION (( 'STEP AP203' ),
                  '1' );
FILE_NAME ('Part3.STEP', '2007-07-11T16:1
FILE_SCHEMA (( 'CONFIG_CONTROL_DESIGN' )
ENDSEC;
-----
DATA;
#1 = ORIENTED_EDGE ( 'NONE', *, *, #31,
#2 = ADVANCED_FACE ( 'NONE', ( #198 ), #
#3 = EDGE_LOOP ( 'NONE', ( #4, #17, #9,
#4 = ORIENTED_EDGE ( 'NONE', *, *, #16,
#5 = MANIFOLD_SOLID_BREP ( 'NONE', #145
#6 = ADVANCED_FACE ( 'NONE', ( #191 ), #
#7 = ORIENTED_EDGE ( 'NONE', *, *, #8,
#8 = EDGE_CURVE ( 'NONE', #15, #11, #189
#9 = ORIENTED_EDGE ( 'NONE', *, *, #10,
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#11 = VERTEX_POINT ( 'NONE', #177 );
```

STEP format

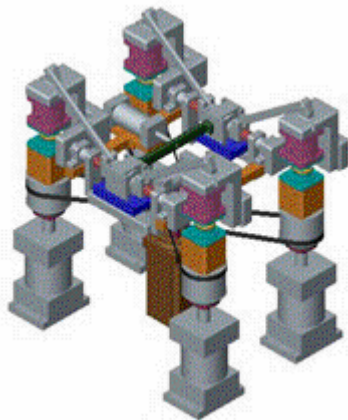
## *Export Options for Transfer*

- *To generate STEP files, different export options are available based on the CAD software being used*
- *Options include solids only, surfaces only, and assembly hierarchy*

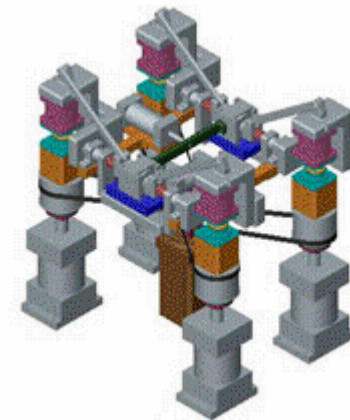
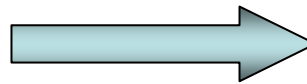


# Complete Transfer of 3D Elements

- *STEP files duplicate the 3D solid and surface geometry of the original CAD model*
- *Line, curve, and point geometry may be included*
- *Coordinates, units, and tolerance stay the same as the original file*



Original CAD  
software

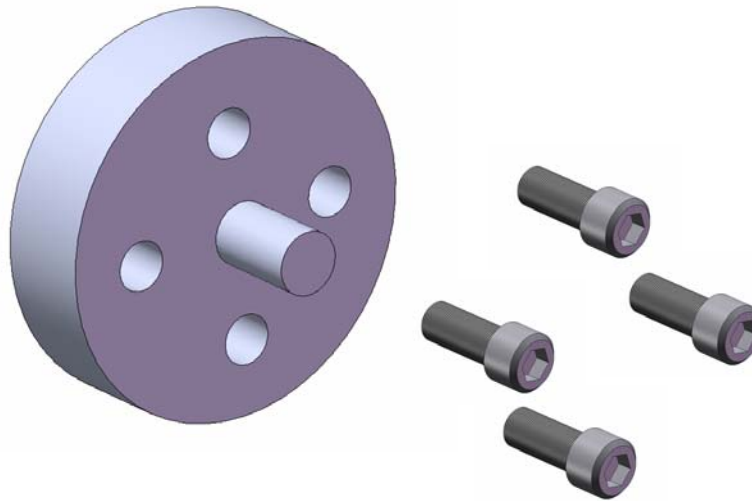


Target CAD  
software

# Grouping Data by Size and Complexity

- A STEP file will typically have a smaller file size than the original CAD file for simple geometry
- Library parts such as nuts and bolts which are all the same and used multiple times can greatly increase file size
- Complex solids will have a larger STEP file size

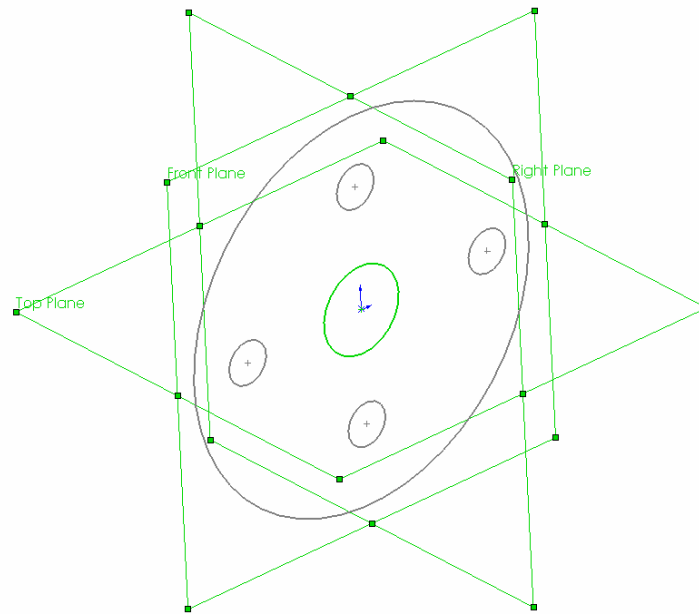
Base geometry  
1 MB



Screw geometry  
8 MB

# Construction Geometry

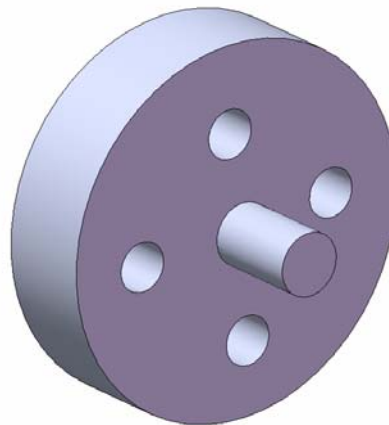
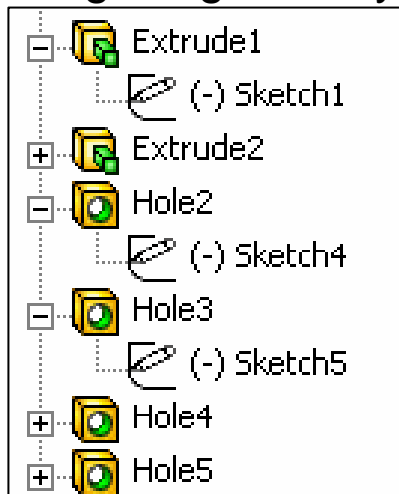
- *When originally created, parts contain construction geometry*
- *Construction geometry includes 2D elements which are extruded, revolved, swept, or lofted to create 3D elements*
- *Construction geometry also includes operations such as cuts, holes, and modifying edges to refine shapes*



# Construction Geometry Issues

- *Construction geometry is displayed in the original CAD application for a user to revert back to a previous configuration*
- *Imported geometry will not include construction geometry*
- *The lack of construction geometry will reduce model functionality*
- *To compensate for a lack of construction geometry, changes can be made to the imported geometry in the receiving CAD software*

Original geometry

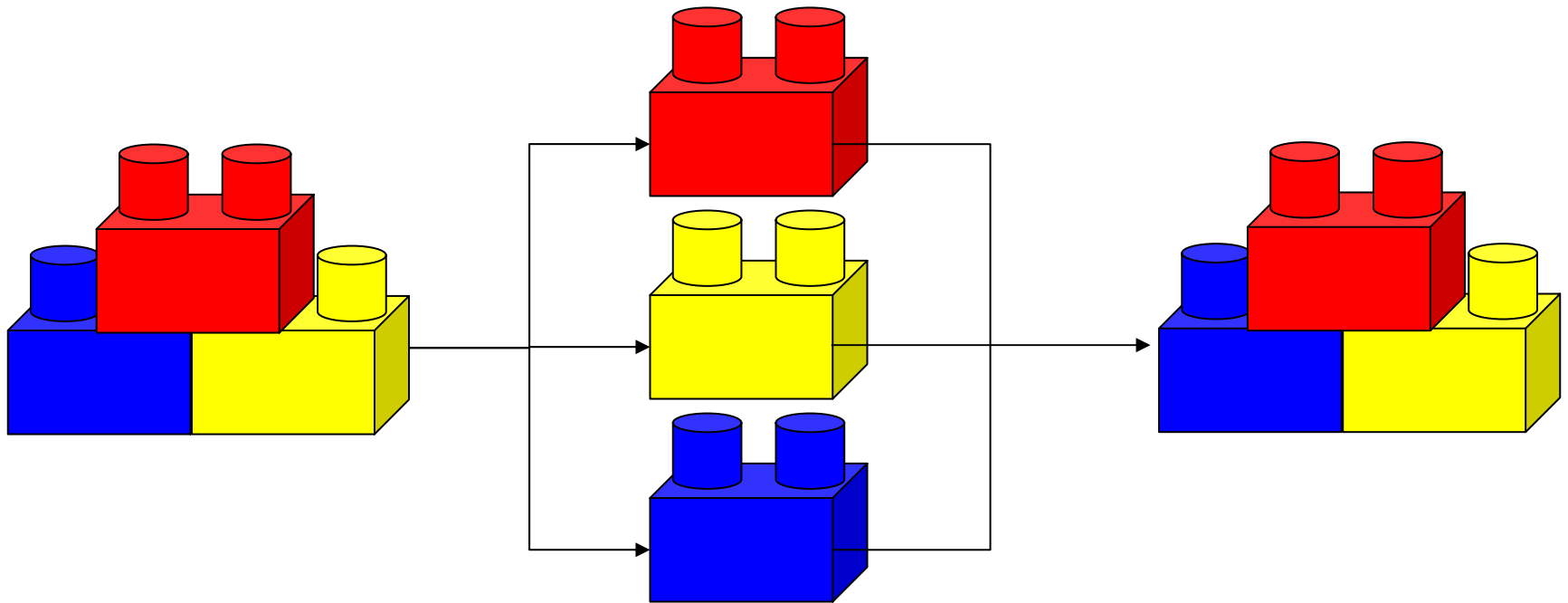


Imported geometry



# Splitting and Merging Assemblies

- *Large assembly files are split into smaller STEP files to be recombined on the receiving system*



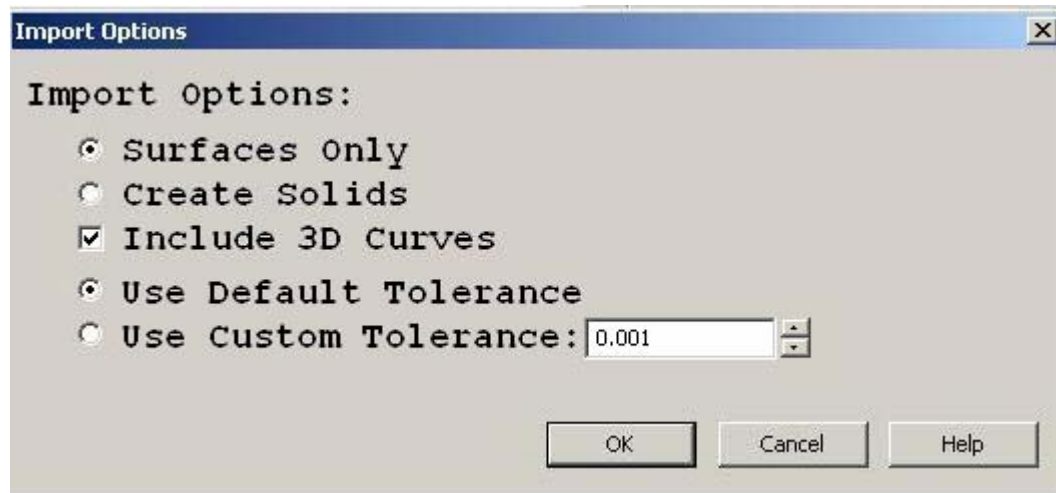
Original CAD  
Assembly of 3 Parts

Parts Converted into  
Separate STEP Files

Imported Parts Recreate  
the Assembly in a New  
CAD Program

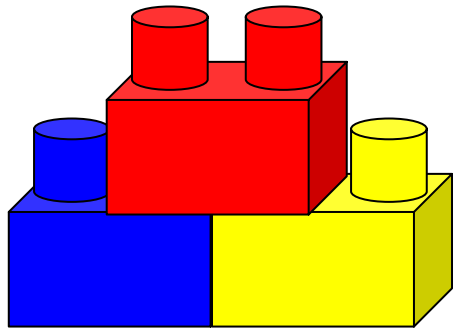
## *Import Options for Transfer*

- *STEP files are imported into a CAD system manually*
- *There are various options for STEP import depending on the software*
- *Options include import only surfaces, create solids, include or omit lines and points, adjust tolerance*

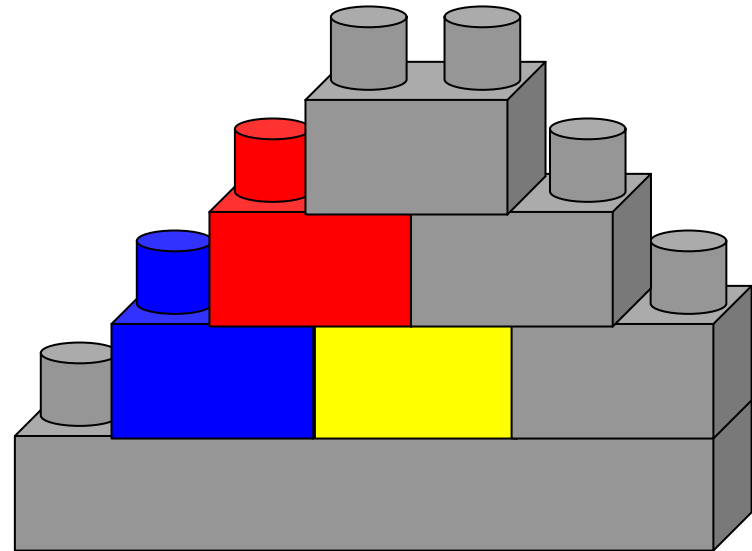


# Importing Model Data into Existing Assemblies

- *STEP files can be imported to create a new part or merged into an existing part*



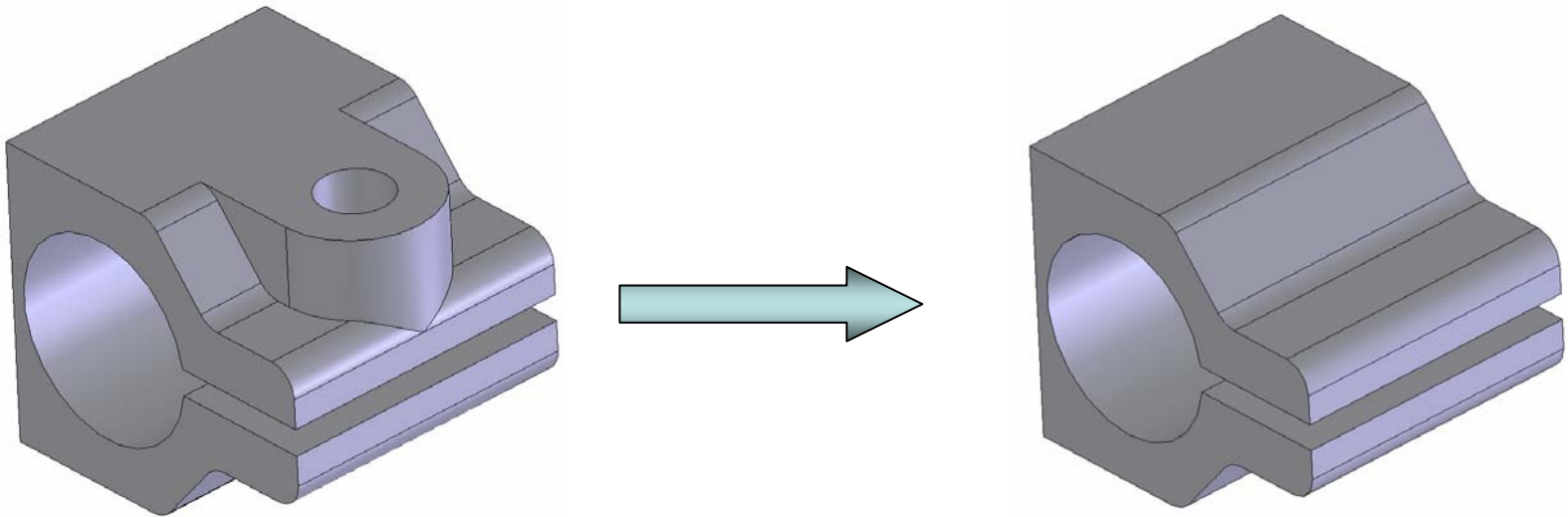
Imported as a New Part



Imported to an Existing Assembly

## Data Transfer Issues

- *Geometry imported from a STEP file may not import correctly*
- *Program messages and logs may not indicate an error during import or export*
- *All data must be visually inspected when imported into another CAD system*

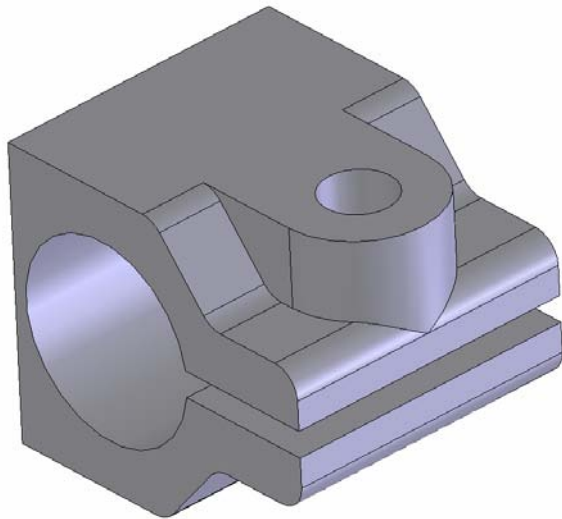


Original Geometry

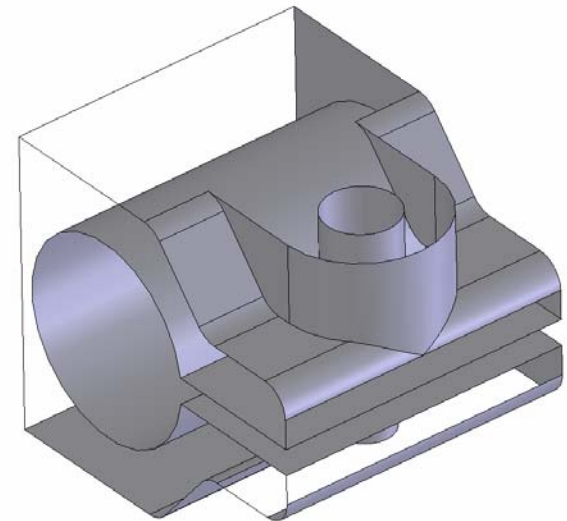
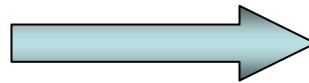
Imported with  
Missing Geometry

## Data Transfer Issues II

- *Solid and surface geometry can be missing or incomplete*
- *Geometry that imports with problems will not have the same number of solids and surfaces as the original model*



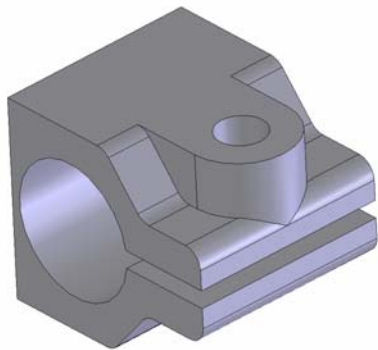
Original Geometry



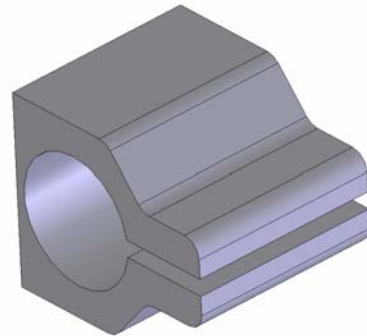
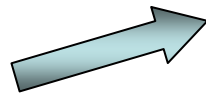
Imported with  
Incomplete Surfaces

## Reworking the Original Data

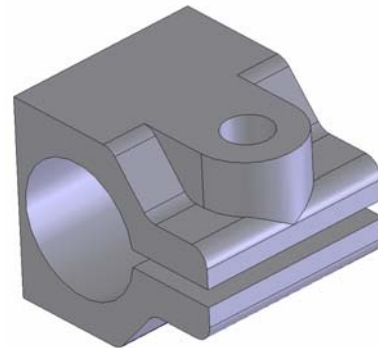
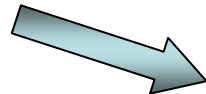
- *All entities will not process correctly*
- *The geometry can be updated to include less complex construction operations*



Original Geometry



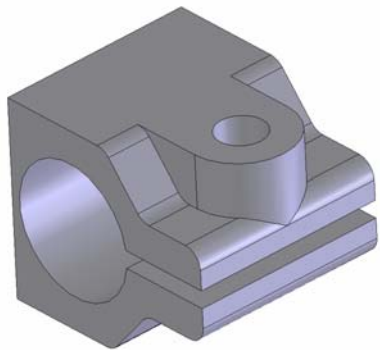
Imported Geometry with Missing Entity



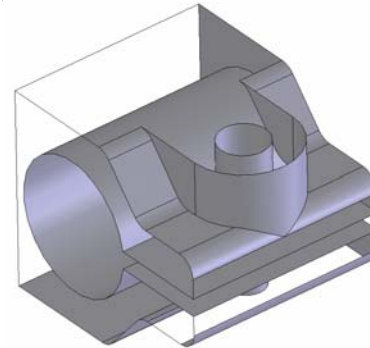
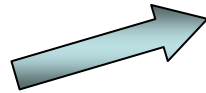
Imported Geometry After Adjusting the Geometry in the Original CAD Software

## Changing Import Options to Fix Issues

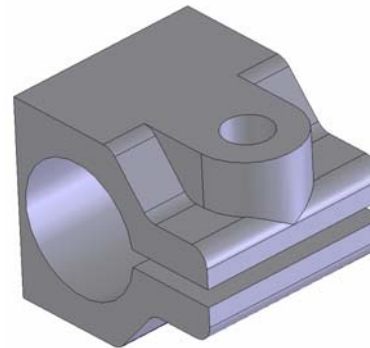
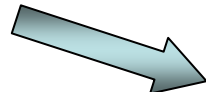
- *Adjusting import options may fix errors*
- *Increasing the tolerance may properly connect boundaries*



Original Geometry



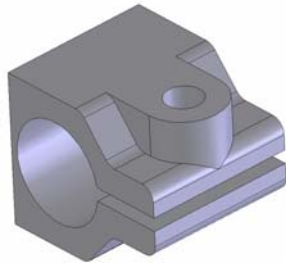
Imported Geometry with Default Tolerance



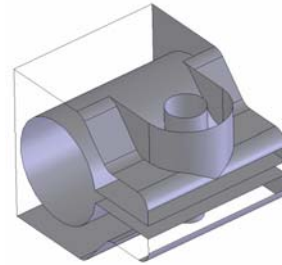
Imported Geometry with Custom Tolerance

# Using Other CAD Software to Fix Issues

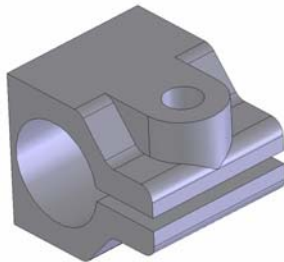
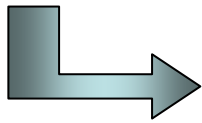
- *Using another CAD program as an intermediate can correct issues*



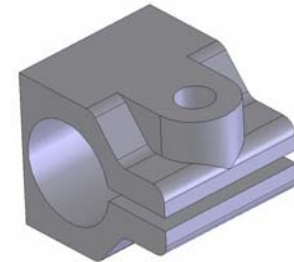
Original Geometry from  
CAD Program 1



STEP File Imported to  
CAD Program 2 with Bad Geometry



STEP File Imported to CAD Program 3,  
Exported as a New STEP File

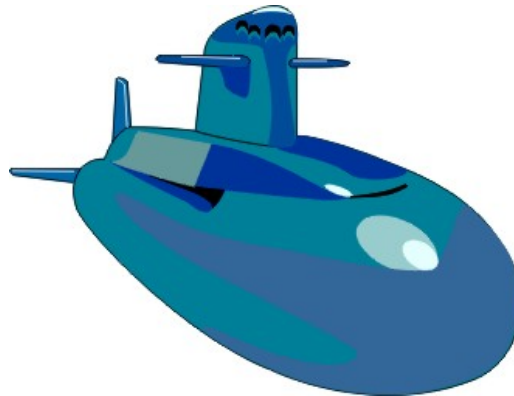


New STEP File Imported to  
CAD Program 2 with Good Geometry

# *The More Complex Systems Become, the More Problems Occur*

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- *Transfer problems occur as a result of the complexity of the original model design or from lack of capability in the export function in the original CAD application*
- *Transfer problems are a daily occurrence when handling the complex assemblies in ship design*
- *By using established procedures, as well as trial and error, nearly all model geometry can be converted*



## *Other Neutral File Types That Are Typically Used for Shipbuilding*

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- *IGES – Early American industry standard for 3D and 2D wireframe and surface geometry exchanges*
- *ACIS – 3D format similar to STEP but not as supported*
- *Parasolid – 3D only format similar to STEP*
- *VRML – 3D format for lightweight visualization*
- *DXF – Exchange format for AutoCAD 3D and 2D transfers*

*STEP does the work of the above formats in a more efficient manner*

## *Advantages of STEP*

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- *STEP files will reliably transfer simple solids and surface geometry*
- *STEP translators are available for most major CAD formats*
- *Other neutral file types such as IGES cannot capture complex shapes without creating files of a huge size*
- *STEP files can be imported and exported into various CAD software to fix bad geometry*

## *STEP Versions are Compatible*

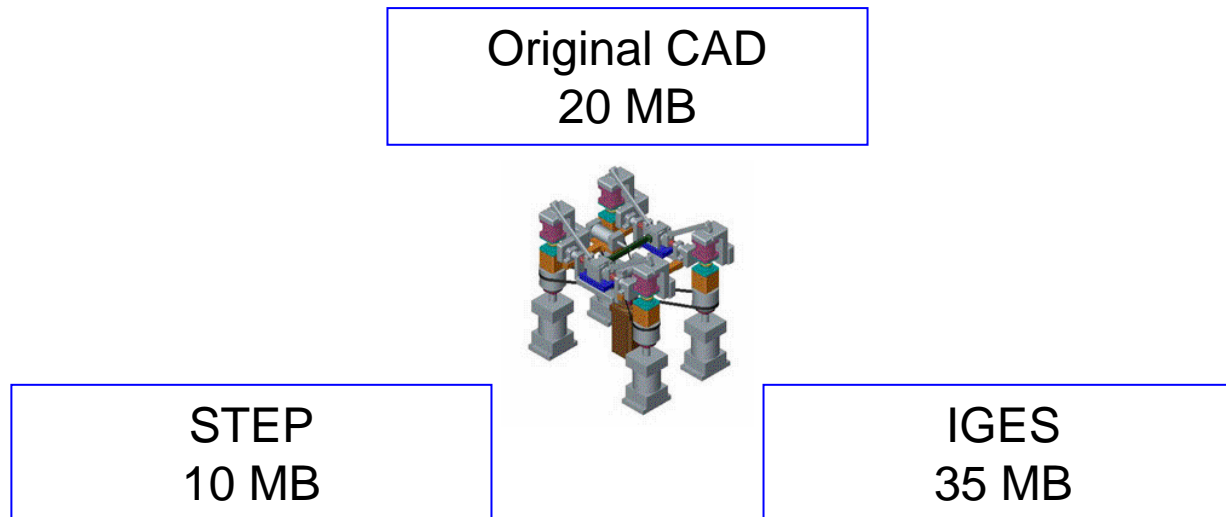
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- *The STEP AP203 format is explicitly upward compatible to future releases and versions of most CAD applications*
- *Most other neutral and proprietary file types have version designators based on a yearly or functionality upgrade*
- *These designators will prevent files from being read by older and newer versions of the software*
- *The STEP format is ideal for file archival since the models will not become obsolete*



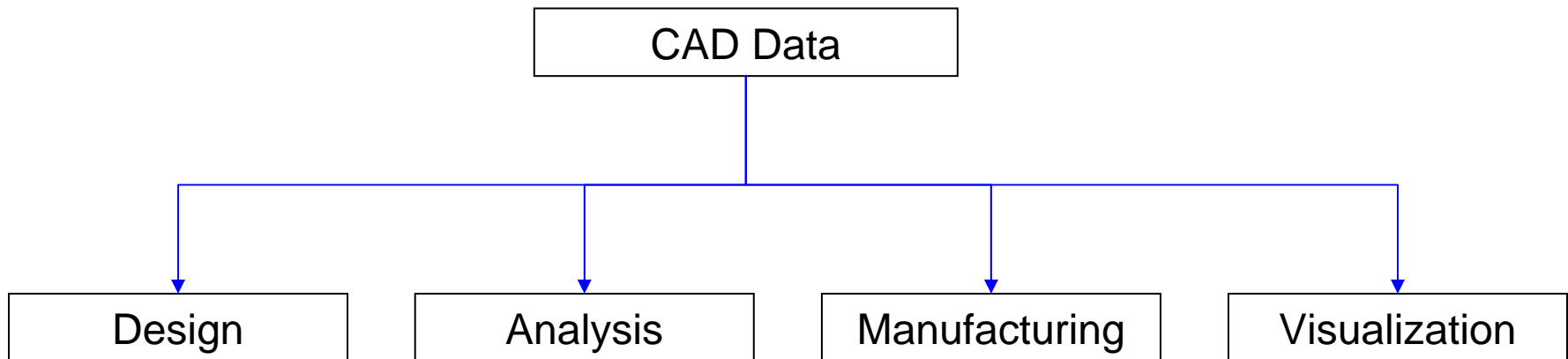
# STEP Files are a Manageable Size

- *For simple geometry, STEP files are a manageable file size*



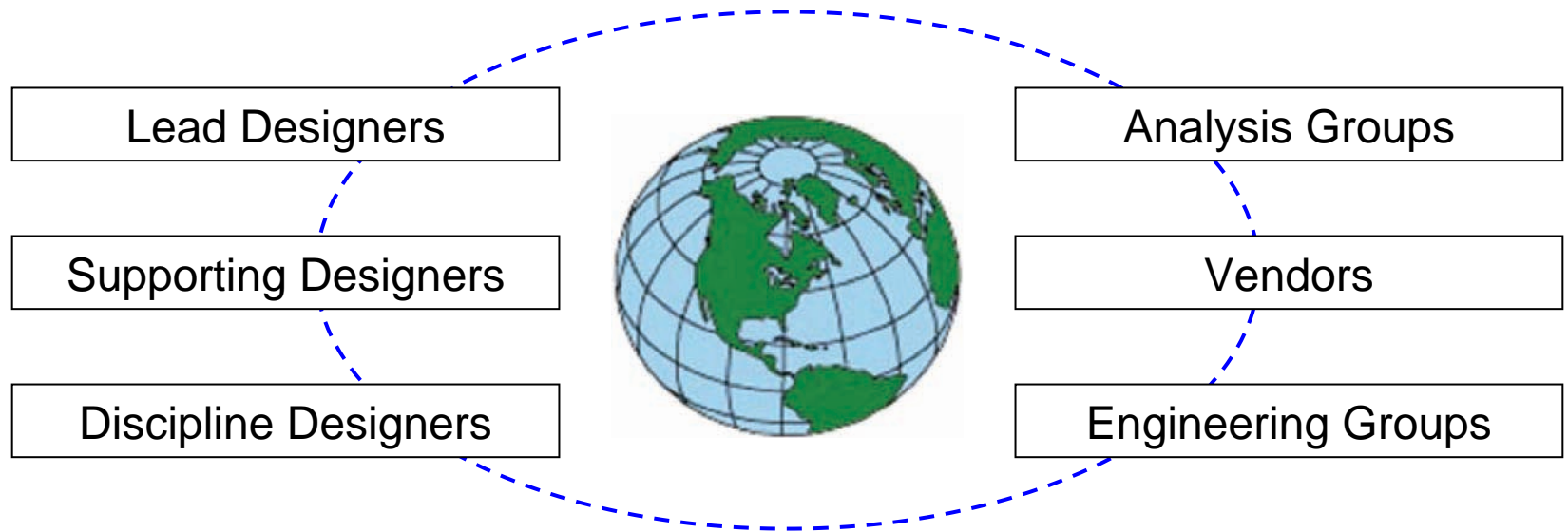
## *A Critical Need for STEP*

- *3D data is expected to be able to integrate into other systems*
- *There are dozens of CAD programs used in shipbuilding, each with different capabilities*
- *Transferring 3D data can rapidly support design, analysis, manufacture, and visualization tasks*



## *Different Groups Share STEP Files*

- *Using STEP is necessary to bridge between different disciplines, tasks, and design partners.*



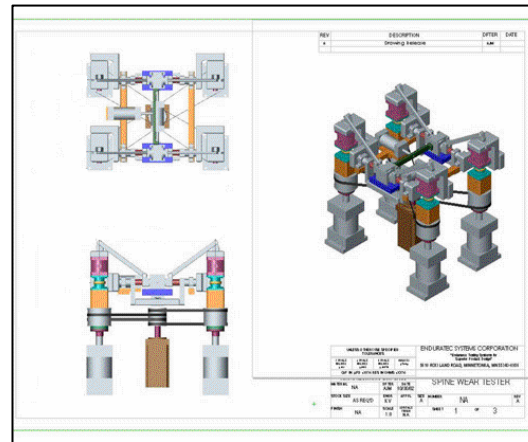
# *STEP Transfer Supports the Use of Multiple CAD Programs*

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- *CAD programs have limitations of maximum model size, visualization options, analysis tools, and manufacturing capability*
- *To get a part's design finalized, it needs to be quickly merged with surrounding data*
- *It is common for different ship systems to be designed on different software platforms*
- *Several import and export iterations are often necessary to work out how a part will fit in with the rest of the system*

## Other Uses of STEP Files

- *For space analysis, interference is checked by importing STEP geometry with existing data*
- *Using software tools, imported STEP data is manually modified to generate views used in 2D drawings*
- *3D STEP data is prepared into a format used to automatically operate manufacturing machines*



# *Different Groups Need the Same Data for Different Purposes*

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- *Using STEP as a transfer medium eliminates the need for a license of every CAD program*
- *Personnel only need training for software their group uses*
- *Utilizing STEP transfer eliminates the need to recreate files*
- *All users have the same data to work from*



# Conclusions

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- *STEP data is a vital and necessary tool in modern ship design and construction*
- *Production data transfer has several neutral file formats, but STEP is the most robust for 3D solid and surface geometry*
- *The requirement to have a variety of design software applications will continue the need for STEP for the next generation of ship designs*

# *Questions and Answers*

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