Student Notes:

CATIA V5 Training
Foils

Healing Assistant

Version 5 Release 19
August 2008

EDU_CAT_EN_HA1_FF_V5R19

About this course

Objectives of the course

Upon completion of this course you will be able to:

- **Analyze the imported data**
- **Repair the imported data**
- **Compare two versions of a Part**
- **Customize the workbench**

Targeted audience

Tooling Designers, Mechanical Designers, Surface Designers.

Prerequisites

Students attending this course should be familiar with the Wireframe and Surfaces.

1 day

Student Notes:

Student Notes:

Table of Contents (1/2)

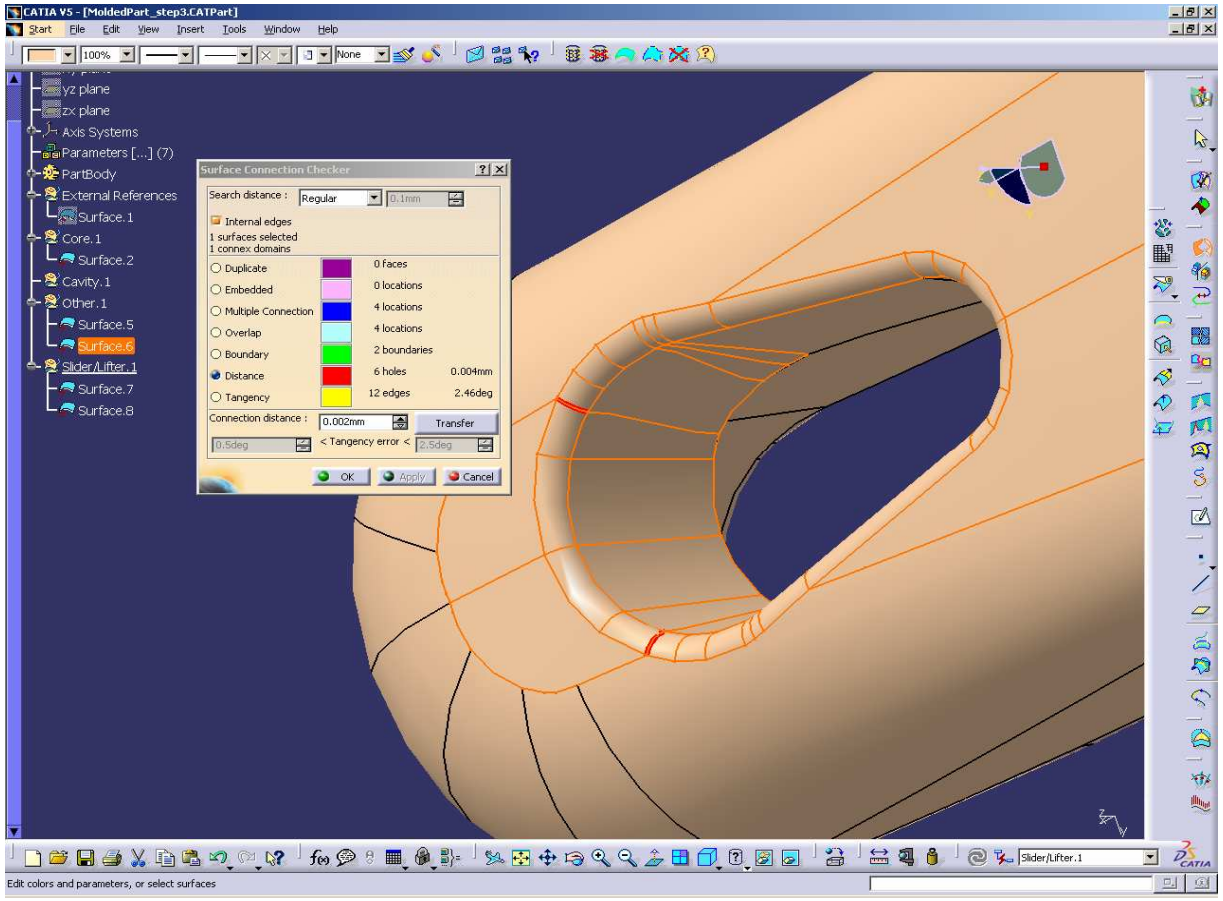
Student Notes:

Table of Contents (2/2)

Student Notes:

Introduction to Healing Assistant

In this lesson, you will see general information about Healing Assistant Workbench.



Student Notes:

Why Do You Need Healing?

Tooling is often done by sub-contractors, who receive data exported by their OEMs in standard formats such as IGES or STEP. In such situations, Healing Assistant allows the sub-contractors to make the Parts "V5 compliant".

Healing Assistant is also very useful for data migration, i.e. for companies who need to migrate data from their previous CAD (including CATIA V4) to their new CATIA V5 environment.

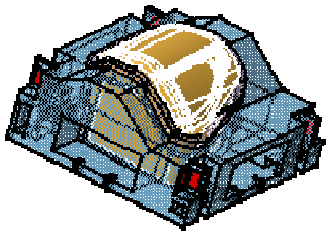
Foreign data: IGES, STEP, CATIA V4, other

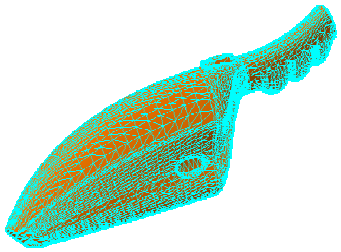
Problems linked to imported data:
- Accuracy uncontrolled
- Validity criteria different from those of V5
- Data quality downgraded during transfer
- User mishandling (losses, duplications)

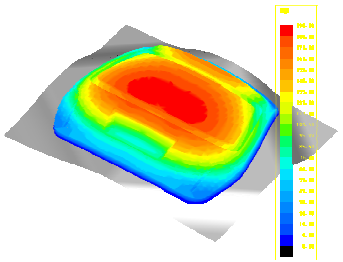
Closed model

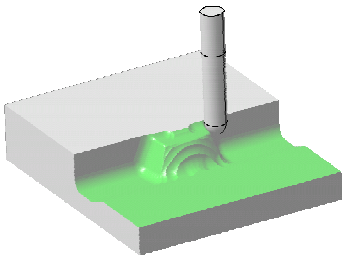
Healing is the process of creating a valid V5 closed model out of any type of data, even not valid.

Tooling Design | Rapid Prototyping | Simulation | NC Manufacturing







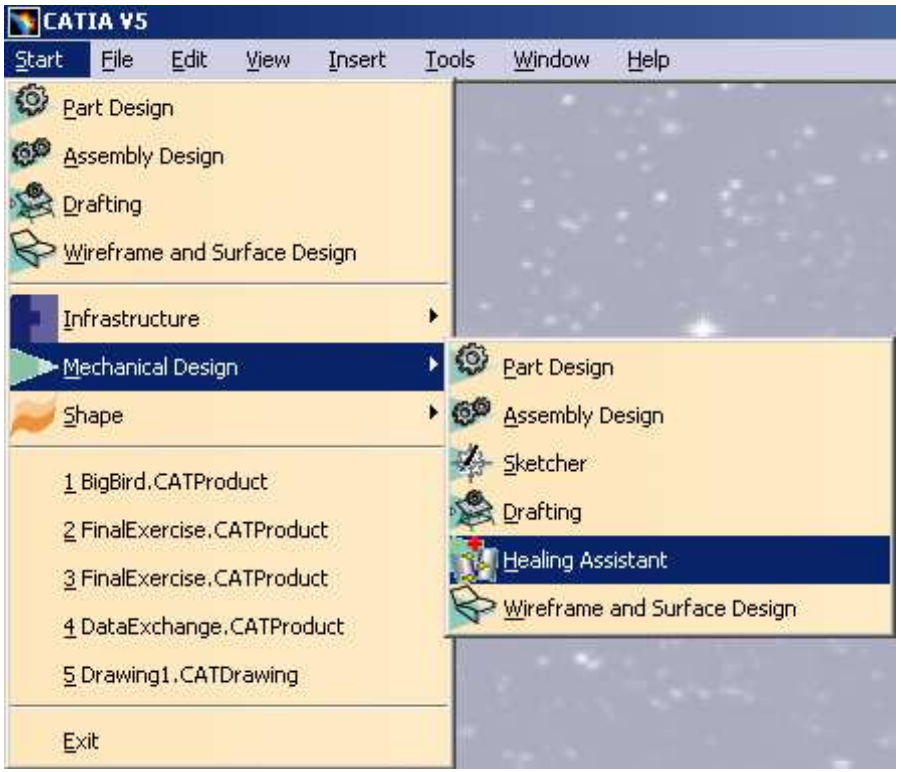


Student Notes:

Accessing the Workbench

To access the CATIA Healing Assistant workbench:

Select Start > Mechanical Design > Healing Assistant.

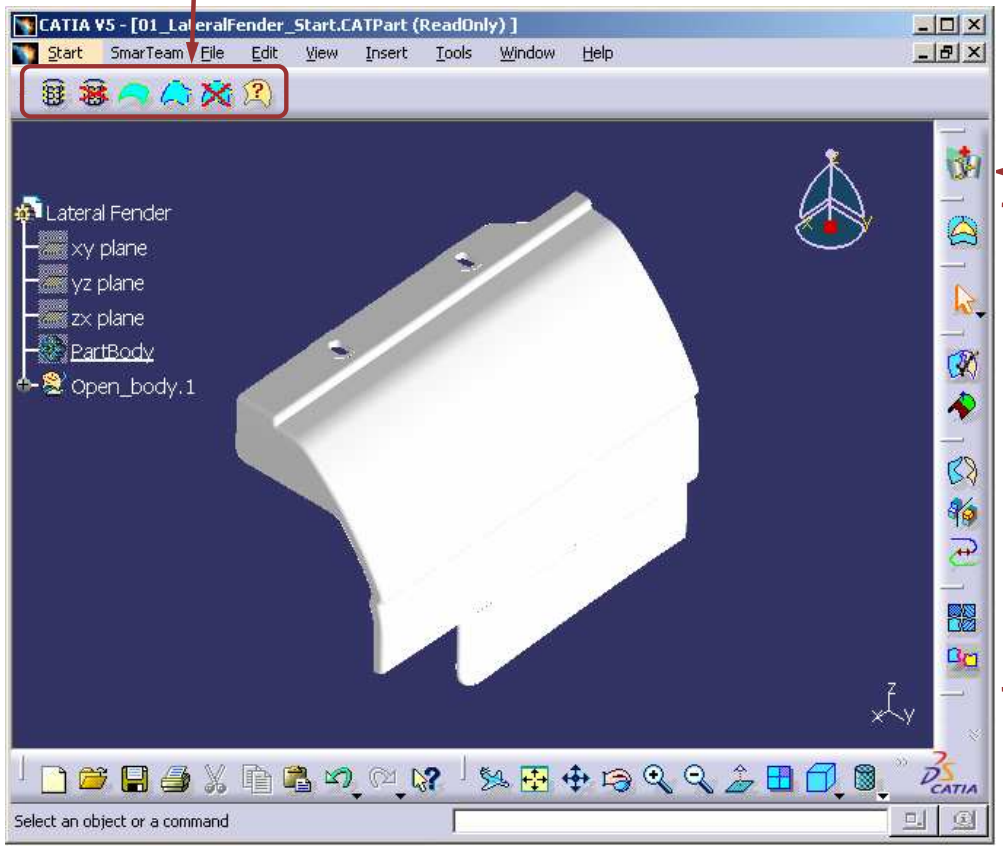


Student Notes:

User Interface: Healing Assistant Editor (1/2)

The user interface of the Healing Assistant workbench is shown below.

This generic tool bar provides access to display modes



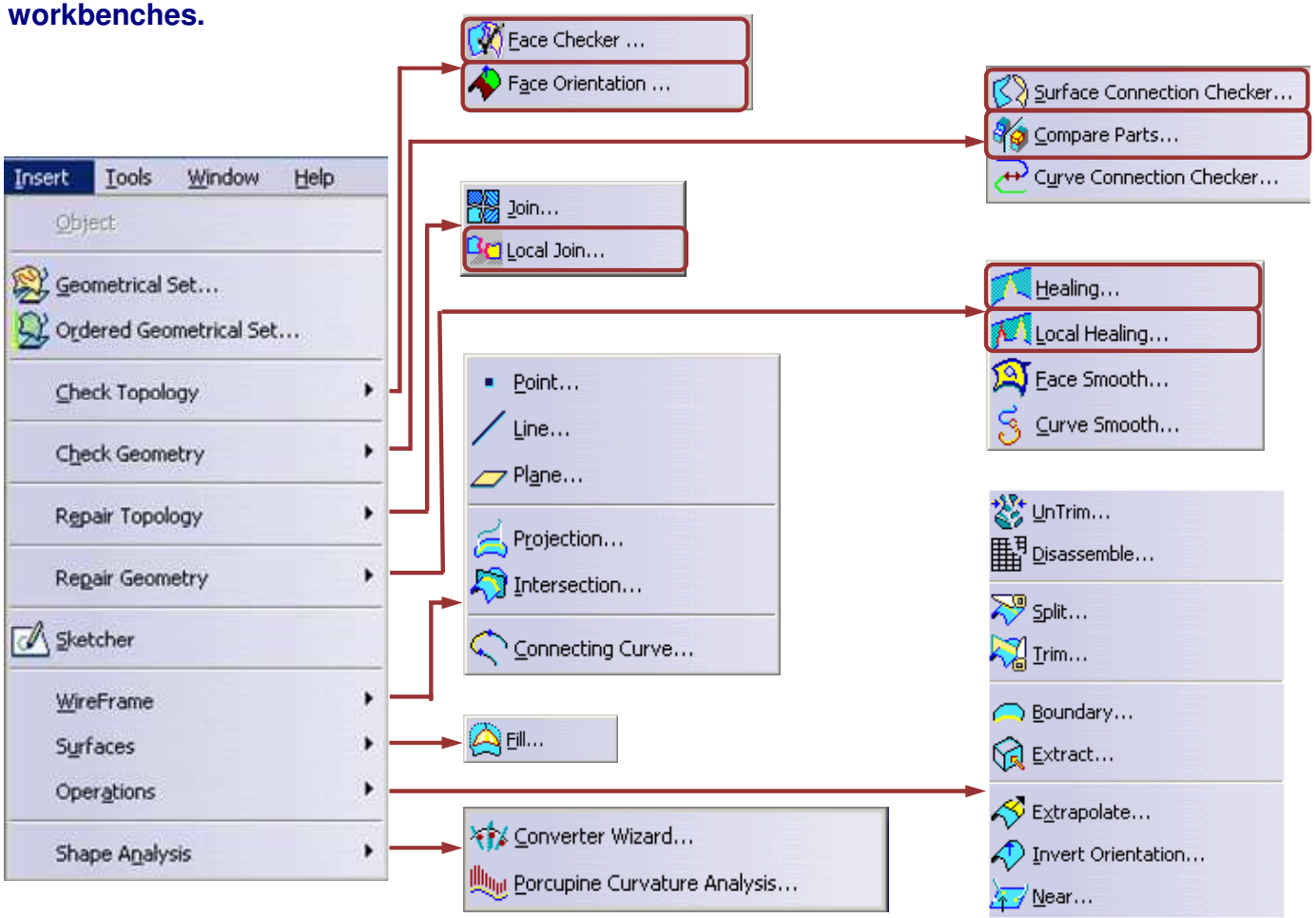
Healing Assistant workbench icon

Digitized Shape Editor Tools

Student Notes:

User Interface: Healing Assistant Editor (2/2)

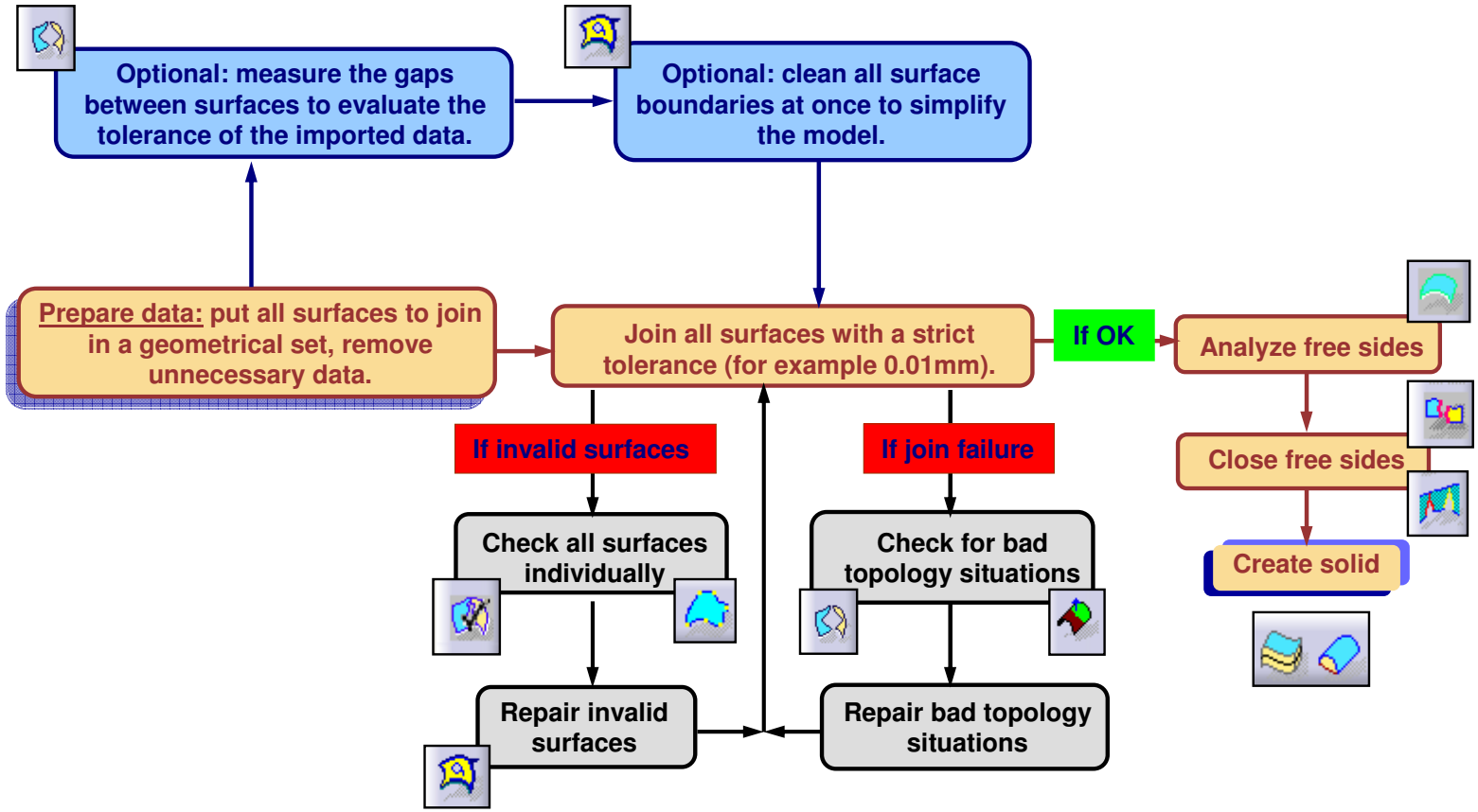
Seven actions are specific to Healing Assistant, the others are taken from other workbenches.



Student Notes:

Methodology For Healing

General process for healing can be seen in the following flow chart.



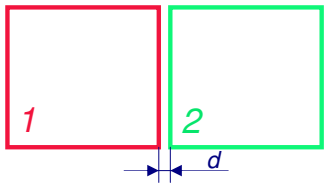
Student Notes:

Information On the Join Operator (1/4)

JOIN is the main operator for the creation of topology. Joining surfaces is done in 2 steps.

STEP 1 is done considering a tolerance parameter called merging distance. All surface boundaries are compared to find those which are close enough to become shared edges.

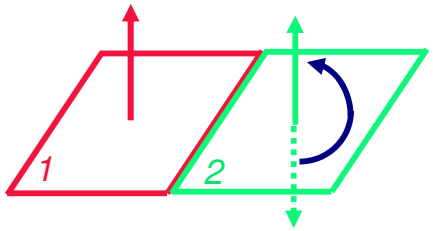
- **d < 0.001mm (system tolerance): edge is shared (geometrically closed)**
- **0.001mm < d < merging distance: edge is shared (only topologically closed)**
- **d > merging distance : edge is not shared, boundaries remain free sides**



PROBLEM: surface boundary curves are not always clean enough to sort out the shared edges.

STEP 2 consists in giving consistent orientations to faces which have a shared edge.

Surface 2 is included to the Join with a flag specifying that the orientation of the face is opposed to the orientation of the original surface (the surface itself is not changed)



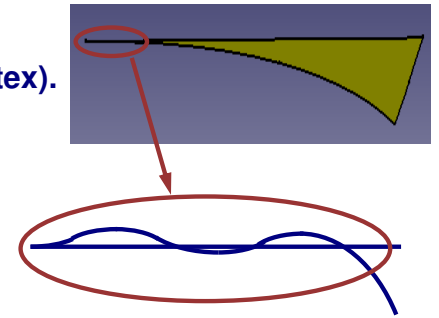
PROBLEM: Because of some invalid situations it may be impossible to propagate a consistent orientation all over the Join.

Student Notes:

Information On the Join Operator (2/4)

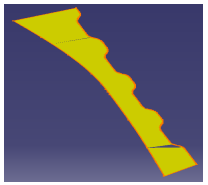
Problem 1: Examples of surfaces with self-intersecting edges

Most common case: surfaces with a thin area.
Depending on the requested merging distance, edges in the thin area are considered as superimposed (impossible to find a single common vertex).

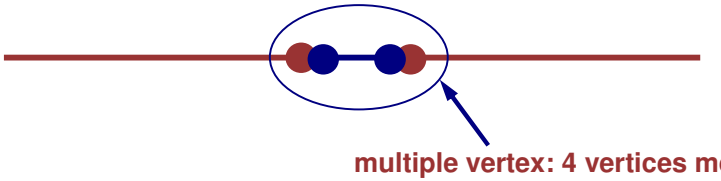


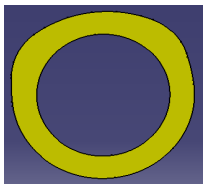
Other situations which may exist in imported data :

- **Incorrect boundary (the shaded display is also incorrect)**

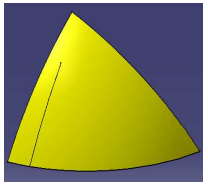


- **Very small curves in the boundary (gives a multiple vertex)**





- **Non supported topology (example: surface lying on a closed surface)**

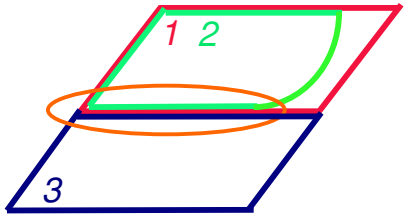


Student Notes:

Information On the Join Operator (3/4)

Problem 2: Examples of Join failures

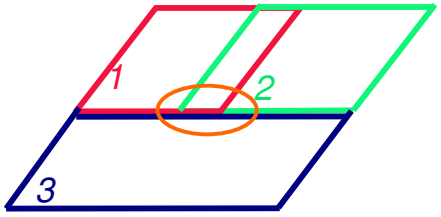
Case of duplicated or embedded surfaces

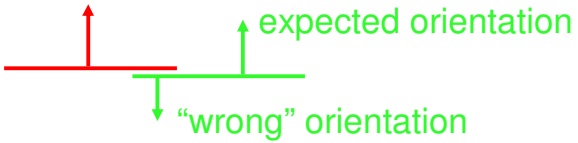


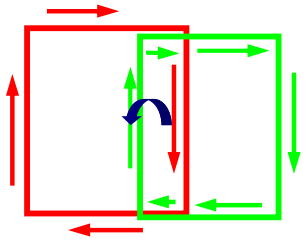
Impossible to find a consistent orientation except if the join operator manages to keep some edges as free sides
In some cases you can get a result, if the Join operator has been able to keep superimposed edges as free edges (boundaries)

Case of small overlap

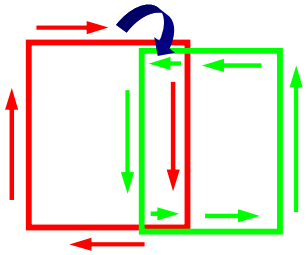
Two possible orientations for surface 2, if the "wrong" one is kept, it will not be possible to add surface 3 to the join.





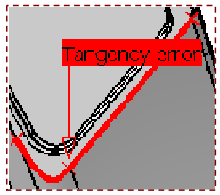


Expected propagation



Incorrect propagation

If possible the Join operator will show the location of orientation inversions; when it is impossible only Healing Assistant can help in finding the defects (usually non manifold edges).



Student Notes:

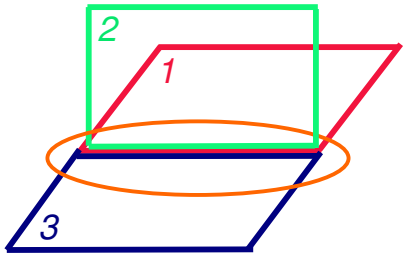
Information On the Join Operator (4/4)

Problem 2: Examples of Join failures

Case of multiple connections:

Standard non manifold edge is edge shared by more than 2 faces.

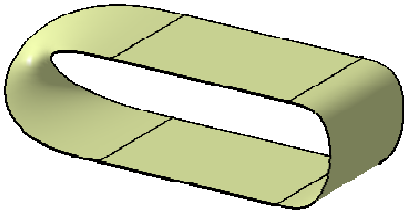
Impossible to find a consistent orientation except if the join operator manages to keep some edges as free sides.



Moebius type situation:

No inside/outside definition is possible.

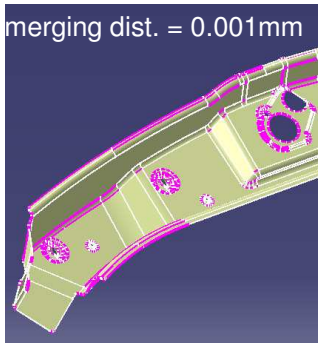
Impossible to define a consistent orientation all over the surface except by keeping a free edge.

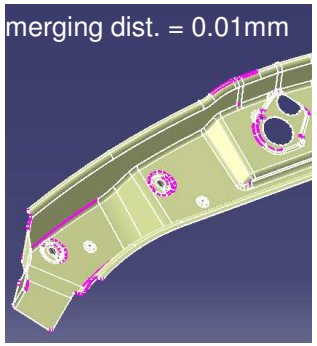


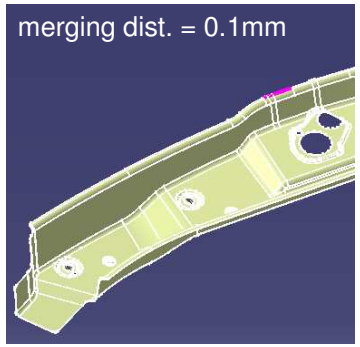
Student Notes:

How To Choose a Merging Distance (1/2)

- You must take into account the accuracy of the data to Join: if you choose a strict tolerance, the Join has many free sides and it takes time to correct them.







- Also sometimes Join does not work with a strict tolerance (Join failure).
- On the other hand when you relax the tolerance, some gaps are hidden but the geometry still has gaps and may be unusable for later processing .
  - For example gaps may cause visible marks on the manufactured part
    remember that a gap may be: or:

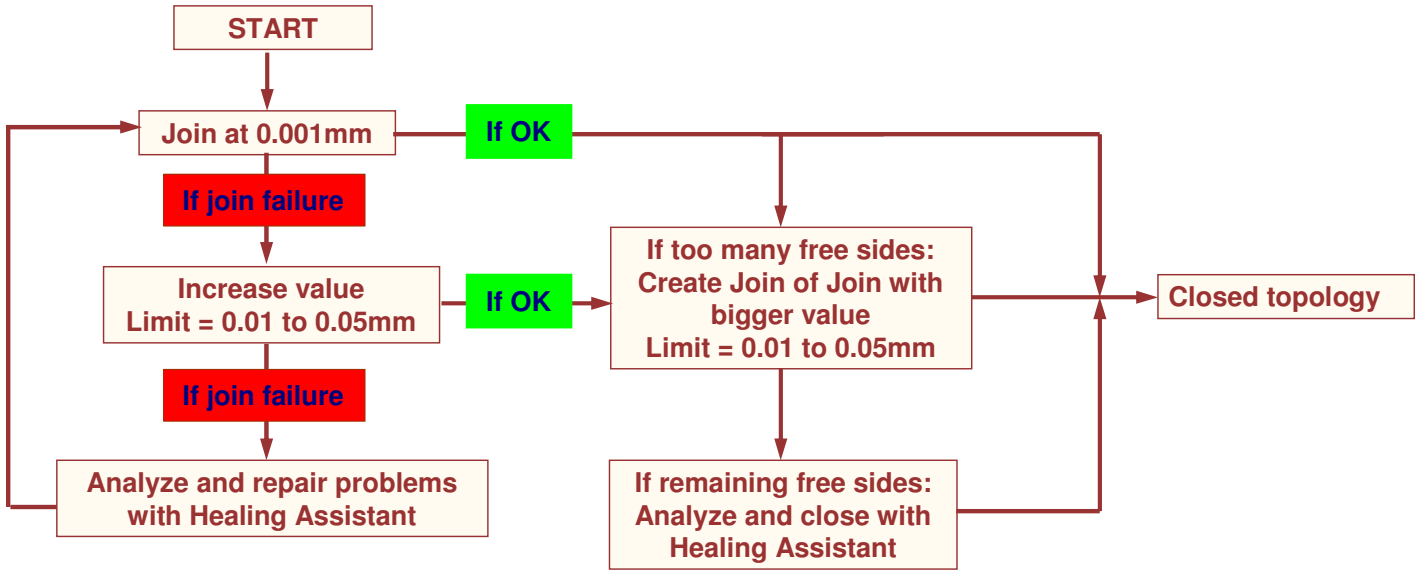
- Finally Join removes all edges which are smaller than the merging distance, which can cause problems if the merging distance is too high.
  - The suppression of small curves may later produce invalid faces (with self-intersecting edges).

Student Notes:

How To Choose a Merging Distance (2/2)

Practical tips:

- Try a Join with a low value of the merging distance (example 0.001 to 0.01mm).
- Increase the value if needed to by-pass failures (example 0.02 to 0.05mm), otherwise use Connect Checker of Healing Assistant to detect invalid topological situations.
- Once you have a Join you may still increase the value of the merging distance to hide gaps; do it by creating a new Join rather than by modifying the first one, in order to avoid suppressing small edge curves.
- Close remaining gaps using Local Join of Healing Assistant.

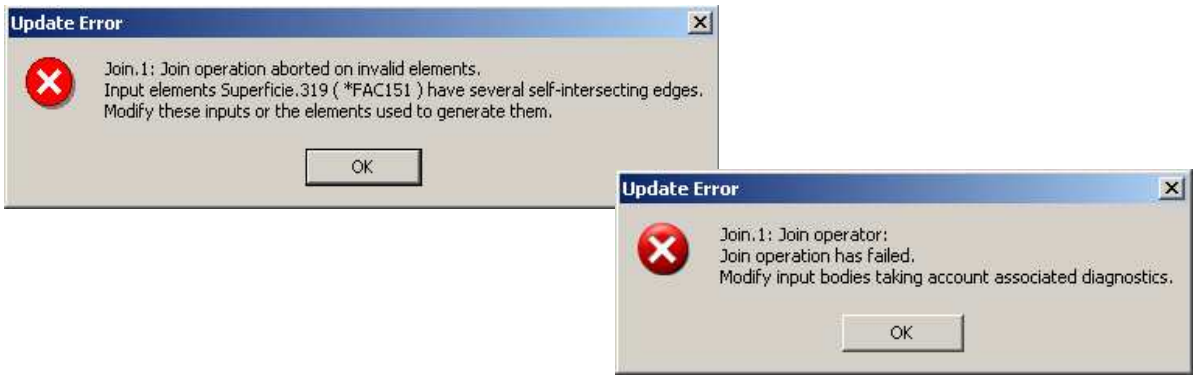


Student Notes:

Model Analysis

In this lesson you will learn about:

- Checking surfaces individually
- Repairing invalid surfaces
- Checking model topology
- Repairing invalid topologies



Student Notes:

Face Checker (1/2)

- The purpose of the face checker is:
  - To find surfaces with an invalid boundary for a given merging distance.
  - To find surfaces with holes in their boundaries and thin surfaces.
  - To sort the surfaces by moving the invalid ones to a new geometrical set.

A. Check internal faces of multi-face surfaces; if the option is not checked only the outside boundaries are analyzed.

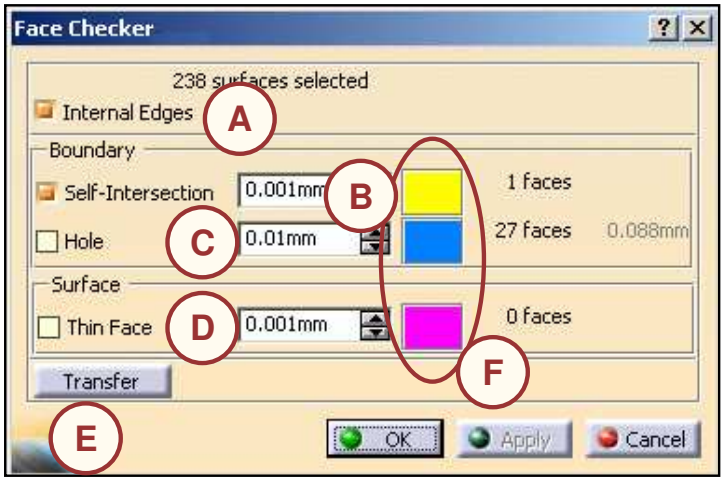
B. Tolerance for the detection of self-intersections: use the merging distance that you intend to use in the Join command.

C. Tolerance for the detection of holes (gaps) in surface boundaries: only surfaces with holes larger than the given value will be highlighted.

D. Tolerance for the detection of thin faces: faces are considered thin when their "width" is everywhere smaller than the specified value.

E. Transfer detected surfaces to a new geometrical set (see next page).

F. Color used for highlight of detected surfaces (you can double-click the color slab to change the color).

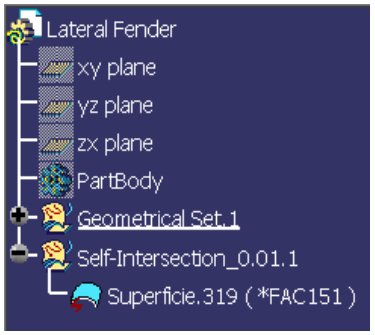


WARNING: checking all surfaces with Face Checker and removing the surfaces detected invalid does not ensure that the Join operator will succeed. New invalidities may appear during the Join process itself.

Face Checker (2/2)

Student Notes:

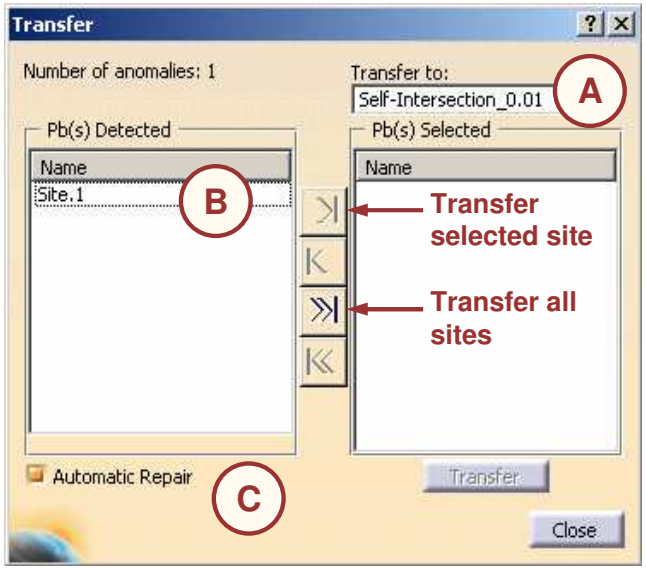
The Transfer button lets you transfer detected surfaces to a new geometrical set.



A. Select a name for the new geometrical set which will receive the transferred surfaces. The new set is created directly under the main part:

B. Surfaces detected invalid for the given criterion (self intersection, hole or thin face). You can select a site to highlight it on screen or to transfer it, you can also right-click to reframe on the site.

C. When the option is checked the operator will try an automatic repair of the detected surfaces.
If it is possible a new surfaces is created in a dedicated geometrical set.
The option is available for self-intersections and holes only = it is not available for thin faces.



Student Notes:

Face Orientation (1/2)

Helpful in case of "Bad Topology" in Join operation. The purpose of the face orientation operator is:

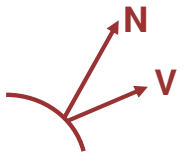
- **To check surface orientation consistency**
- **To invert the surfaces with a wrong orientation**

A. Options:

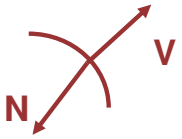
1. **While Moving: the colors change dynamically when the view direction changes.**
2. **Direction locked: you can change the view while keeping the same direction for orientation analysis.**
3. **Fly analysis: when direction is locked you can display orientations on the fly.**

B. You can reverse the orientation of a given surface or propagate the orientation of a given surface (see next page).

C. Each surface is displayed with one of the colors, depending on the direction of its normal direction (N) compared to the view direction (V). Colors can be changed by double-click on the color tabs.

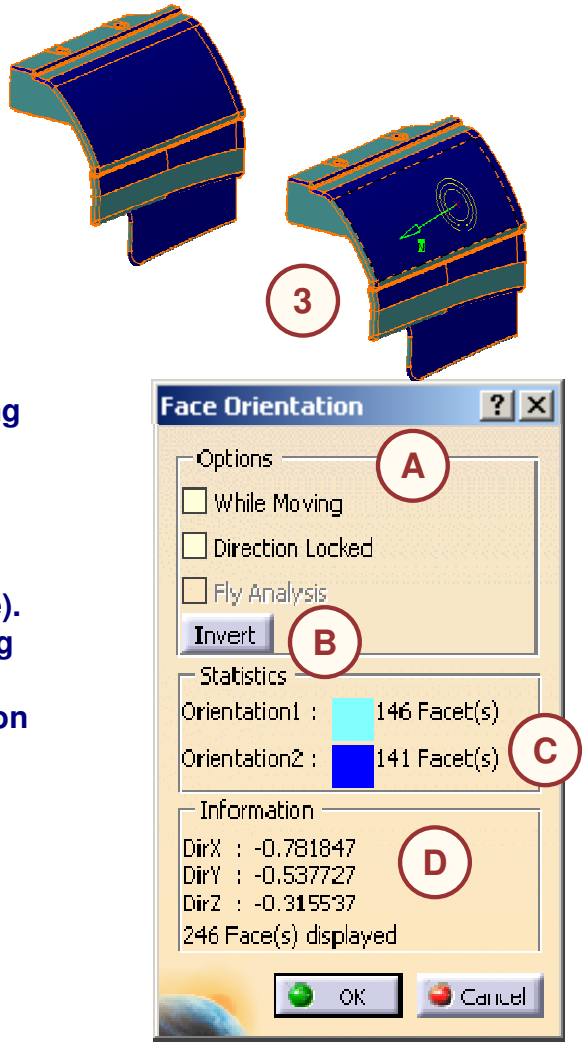


Orientation 1
N.V >0



Orientation 2
N.V <0

D. Information: current view direction and number of selected faces.

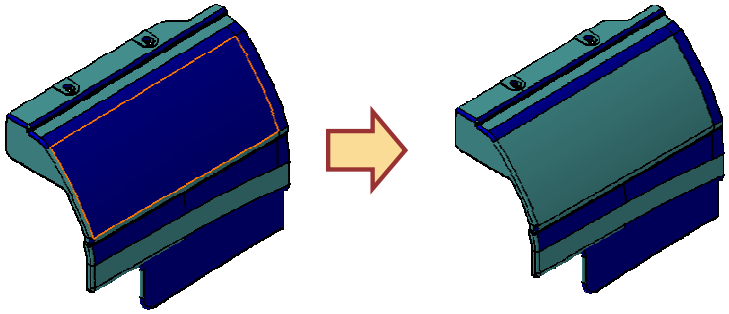


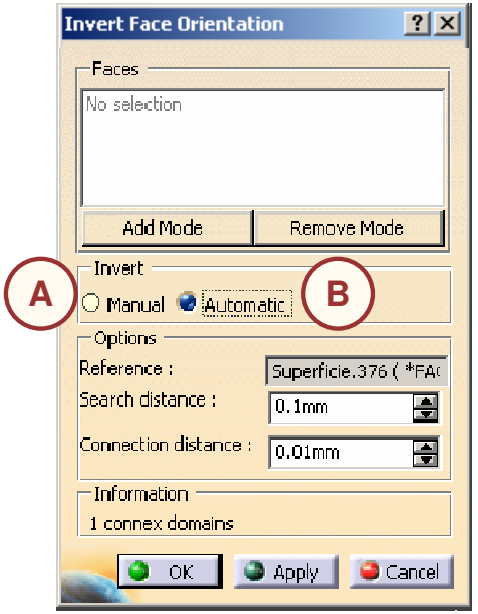
Student Notes:

Face Orientation (2/2)

Invert surface Invert

A. Manual: Only some selected surfaces are inverted.



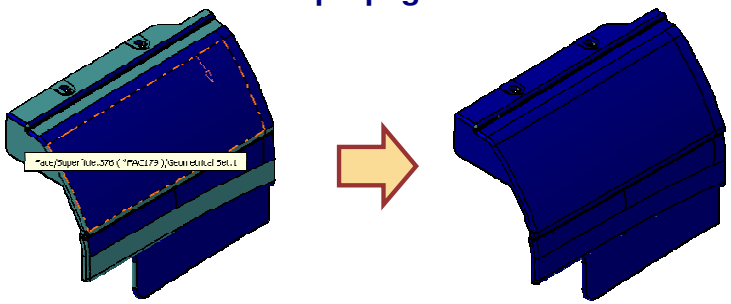


B. Automatic: The orientation of a selected surface is propagated to all other surfaces of the selection (consistent orientation).

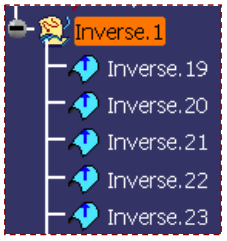
a. Options: same as Connect Checker.

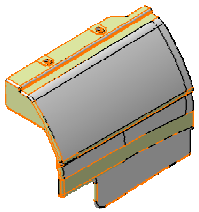
b. Search distance: distance used to pre-select boundary edges that will be compared (edges candidate for merging).

c. Connection distance: distance below which two candidate edges are actually considered as merged and the orientation is propagated.



The processed surfaces are hidden and the new surfaces (with inverted orientations) are created in a new Geometrical Set.



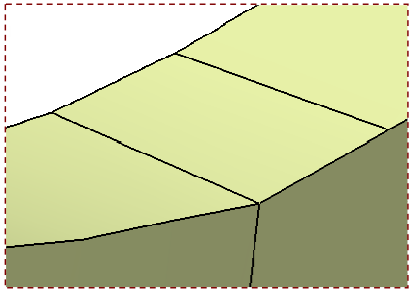


Student Notes:

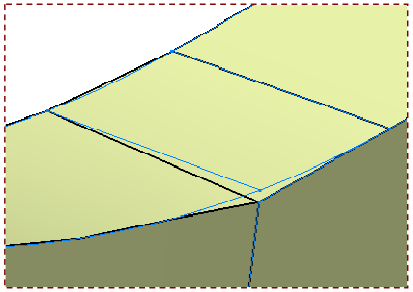
Geometrical Display

The purpose of the Geometrical display is:

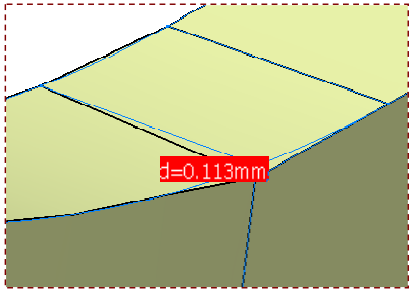
- To check surface geometric boundaries visually (remember that the regular surface display shows topological boundaries).
- To detect holes in surface boundaries.



Regular shaded display

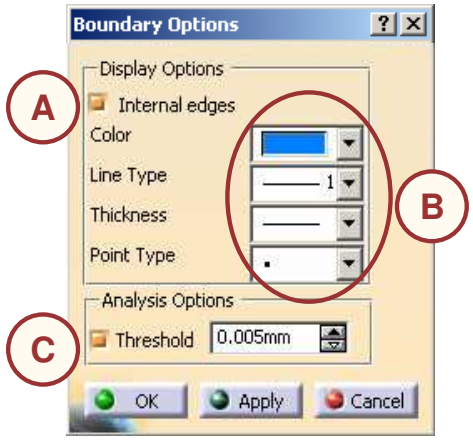


With geometric boundaries



With geometric boundaries + holes

A. Internal edges: you can display the internal boundaries of a multi-face surface.
B. Graphic properties for the display of boundaries.
C. Threshold for the display of holes in boundaries: only holes wider than the specified value are highlighted.



Use Remove Geometrical Display to turn OFF the display of surface geometric boundaries.

Student Notes:

Repairing Invalid Surfaces

- You can repair invalid elements by one of these methods:
  - Recreate by Split using the boundary and support surface of the invalid surface.
  - In particular cases, it may be quicker to discard the invalid surface and to create a new one using the standard surface creation tools of CATIA.

General case: recreate the surface by SPLIT

Isolate the invalid element
Create its full BOUNDARY (as a datum)

If necessary:
SMOOTH CURVE to the boundary

DISASSEMBLE the boundary,
check and repair the curves and

JOIN them back

UNTRIM the surface

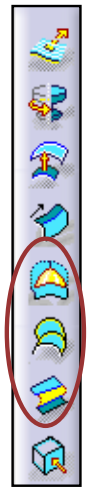
SPLIT the surface back.

Simple situations: replace the surface by a new element

FILL

MULTISECTIONS SURFACE or

BLEND (specially for ruled surfaces)

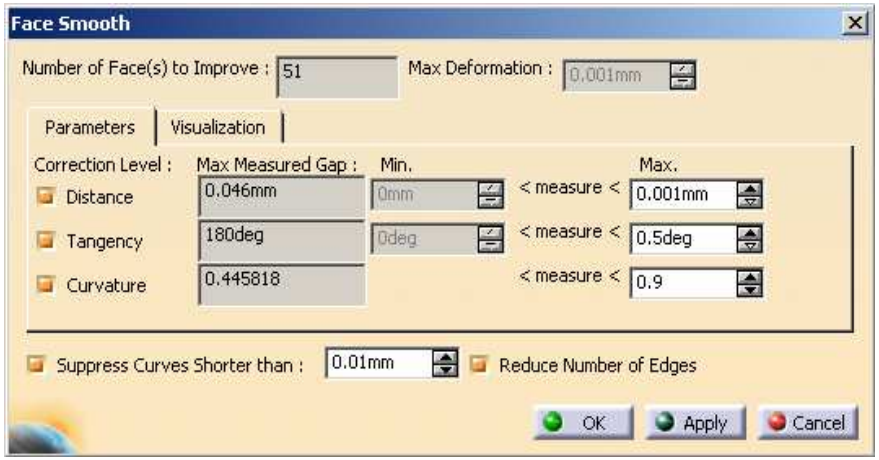


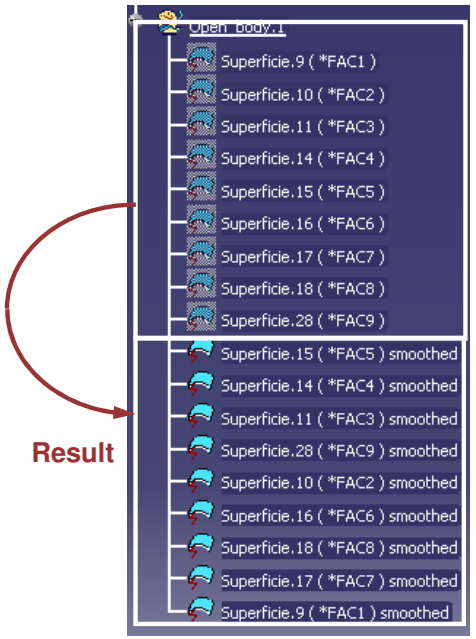
Repairing the invalid elements is usually very easy and quick once they have been located.

Student Notes:

Face Smooth (1/2)

- Invalid surfaces can also be repaired by the Face Smooth command (mono-face surfaces only).
- It modifies the edges of the boundaries within a tolerance in order to:
  - Fill gaps between edges up to a given length.
  - Correct small tangency breaks up to a given angle.
  - Correct curvature discontinuities up to a given % value.





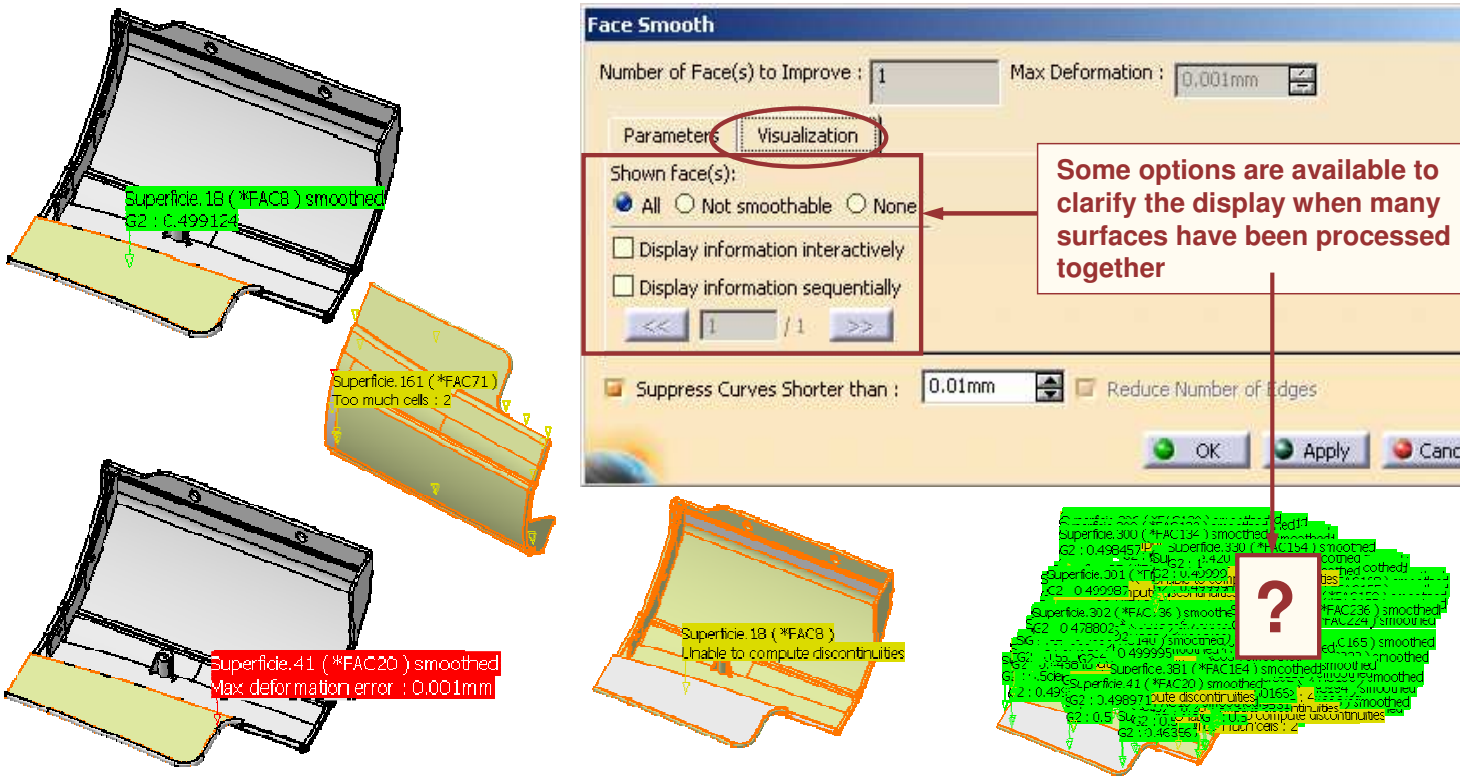
- It can also
  - Suppress very small edges by removing them or concatenating them.
  - Reduce the number of curves by concatenating edges which are continuous in tangency.

Student Notes:

Face Smooth (2/2)

A Visualization tab allows to have a feedback on the result. It shows:

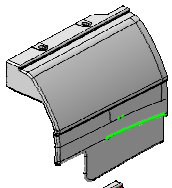
- **The remaining discontinuities between boundary edges (green)**
- **The surfaces which could not be processed (yellow)**
- **The surfaces where the deformation is greater than the system tolerance (red)**



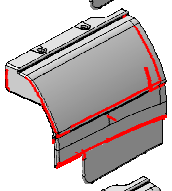
Student Notes:

Surface Connection Checker (1/2)

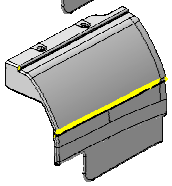
- You can analyze the connections between several surfaces or inside a multi-cell surface (internal edges).
- You can highlight discontinuities greater than a given value.
- You can highlight duplications, overlaps, embedded elements…



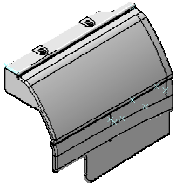
Boundary: cannot be merged with any other edge



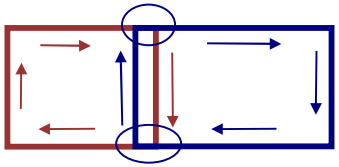
Distance: geometric gap (between connection and search distances)



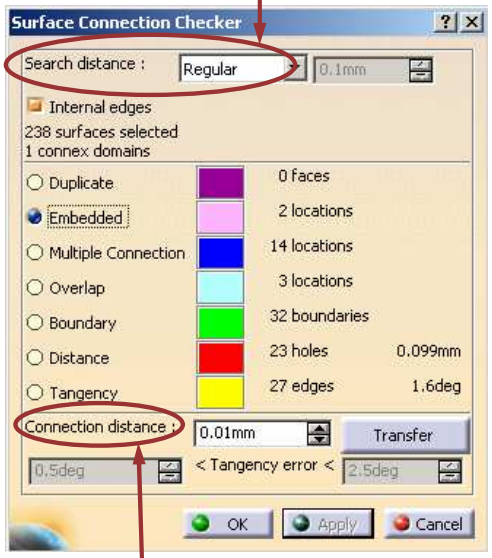
Tangency: discontinuities along merged edges according to Tangency error criterion



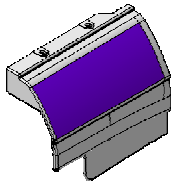
Overlap: merged edges with no possible consistent orientations



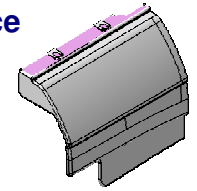
Parameter used for pre-selection of edges:
If d < Search distance, a more accurate test is done to find if edges can be shared



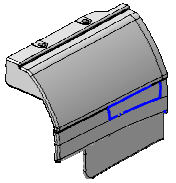
Tolerance that you intend to use for Join.



Duplicate: same area with same edges



Embedded: a surface is totally included into another one



Multiple connection: more than two edges can be merged together

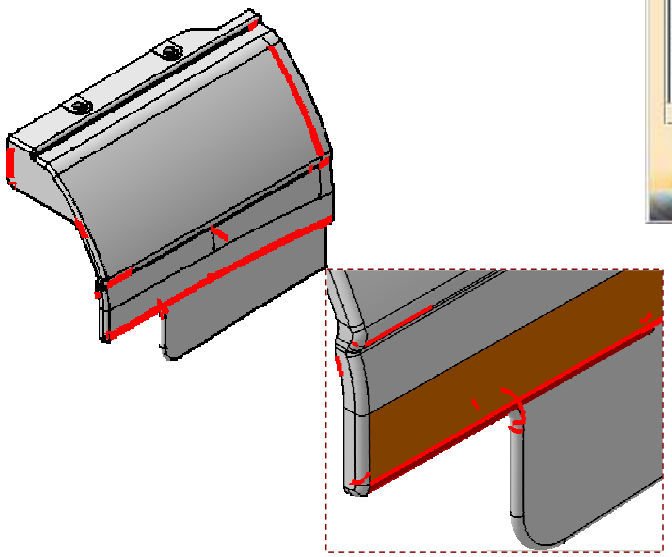
Student Notes:

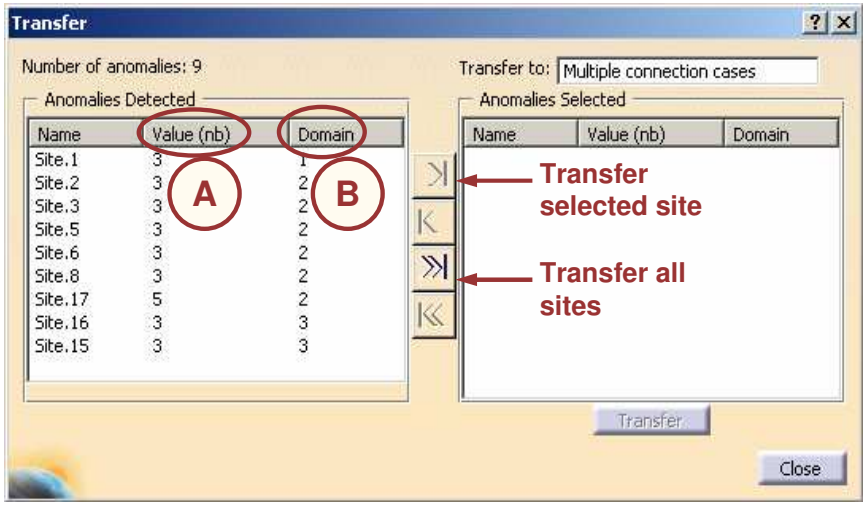
Surface Connection Checker (2/2)

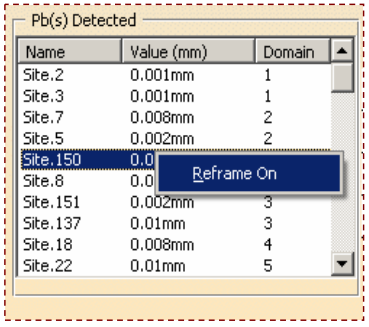
- Discontinuities are identified as sites grouped into connex domains.
- You can highlight a site and use Reframe to analyze it.
- You can transfer sites to a new geometrical set for later analysis.

A. Domain: domain number for the selected site

B. Value: distance, angle, number of merged edges depending on the type of analysis performed



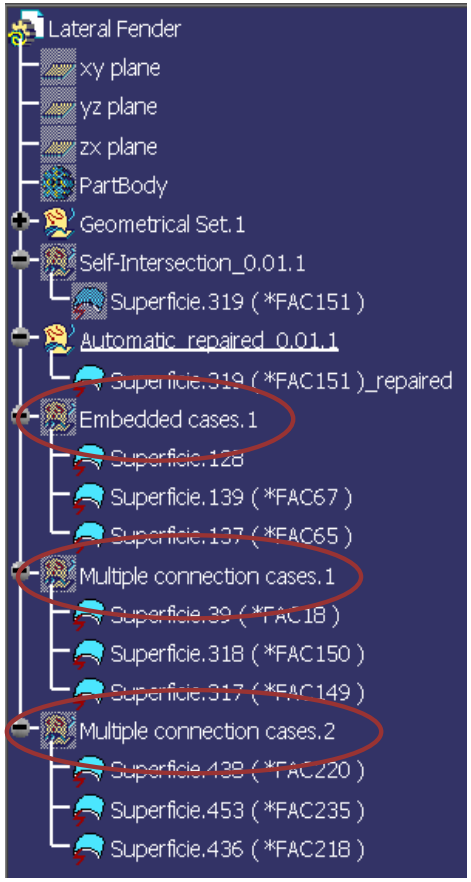




Student Notes:

Repairing Topology

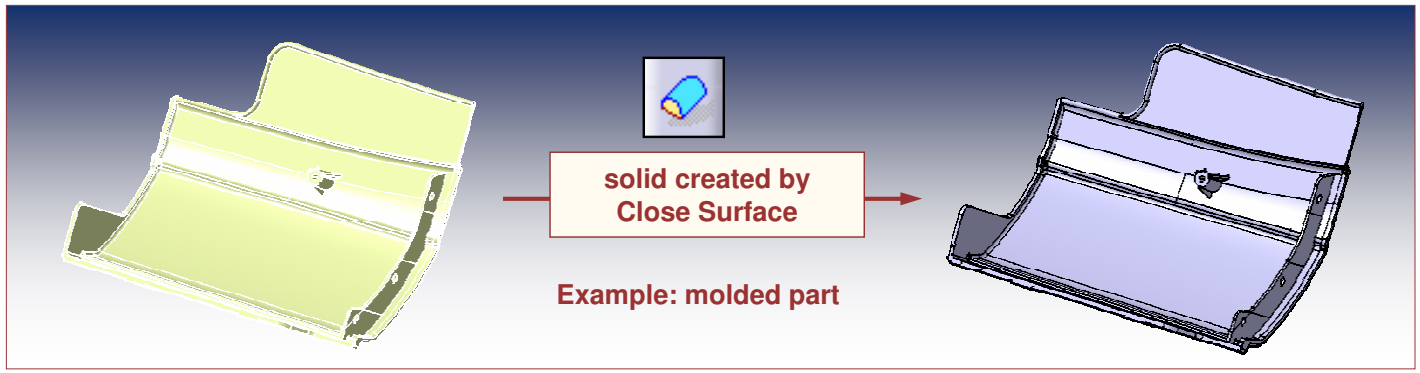
- Surfaces identified as possible causes for bad topology situations have to be sorted out using the connection checker.
- These surfaces must be carefully analyzed to find out if they cause a problem or not.
  - They may simply be adjacent surfaces with opposite orientations.
- Bad topology situations can usually be repaired by:
  - Deleting unnecessary surfaces
  - Trimming surfaces
  - Correcting surfaces which cause an inconsistency in orientation
- Repairing bad topology situations is easy once they have been precisely located on the model.

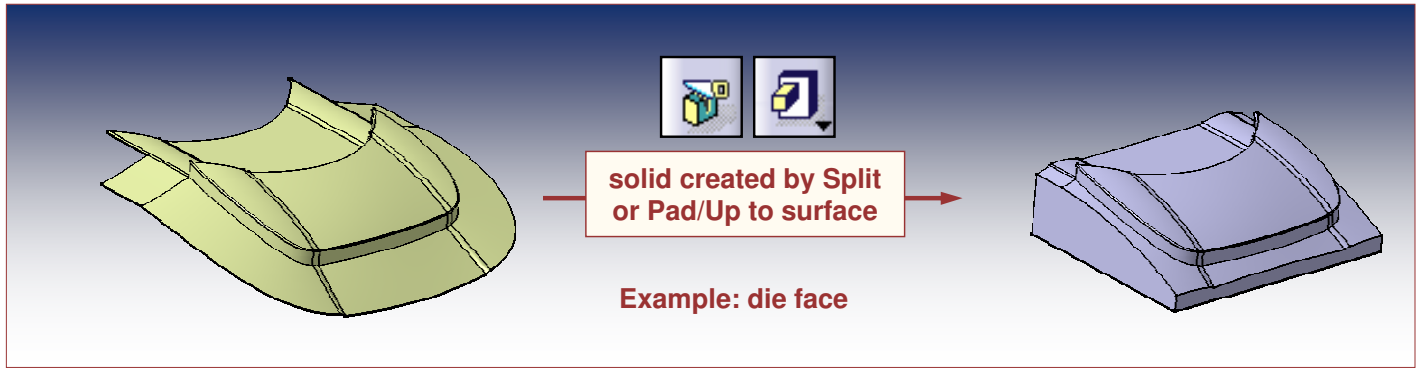


Student Notes:

Closing a Join Surface

Once you have created a topology you have to make it closed (watertight, seamless).
For that you need to analyze and suppress all free sides.



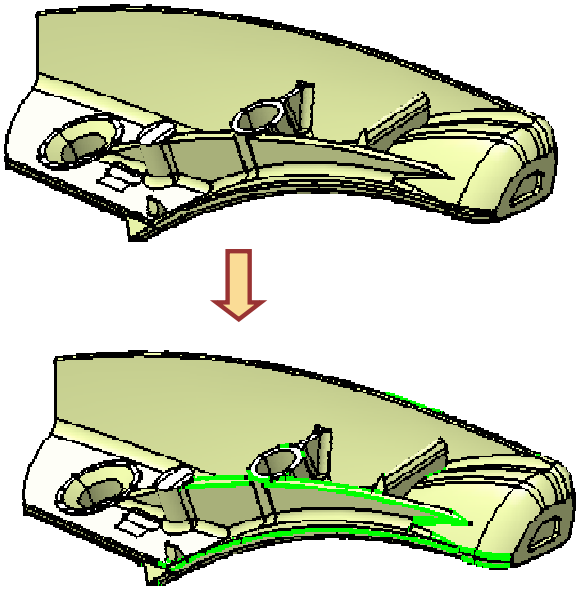


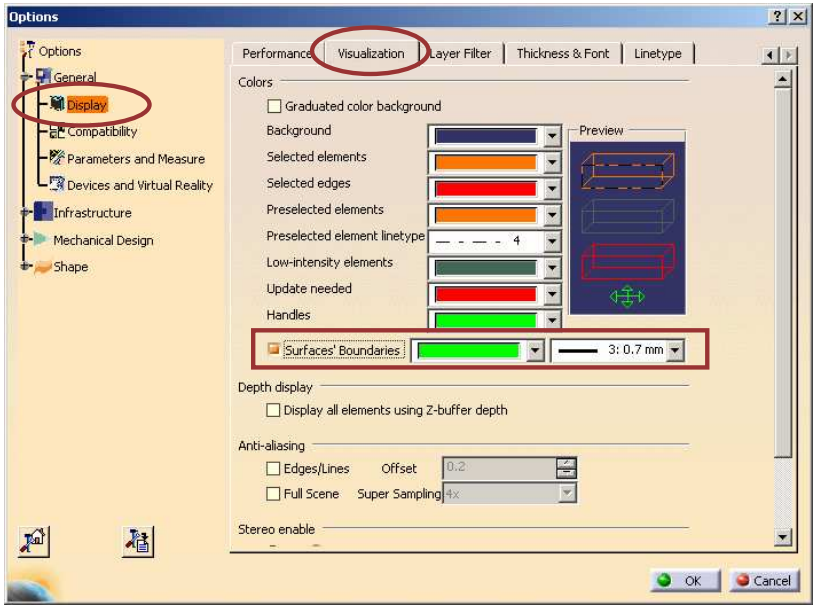
Checking For Free Sides (1/2)

- You can display the free sides of a surface with the Surface Boundaries icon of Healing Assistant

  - You can also use the Boundary function to highlight the free sides.
    The Boundary function gives an explicit message when the surface is closed.

  - Or use the Tools/Options/Display/Visualization menu
    The option tool box allows to choose the color and thickness of boundaries to display.



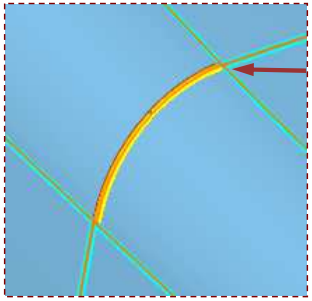


Student Notes:

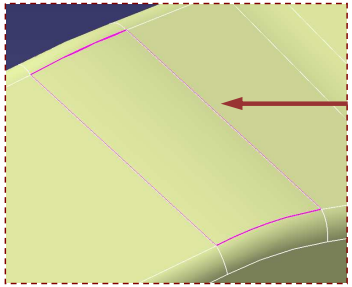
Checking For Free Sides (2/2)

Student Notes:

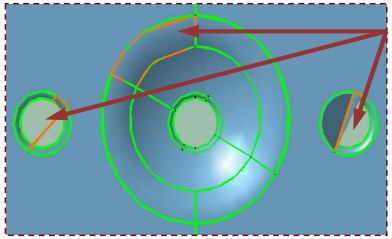
There are only 4 possible cases of free sides.



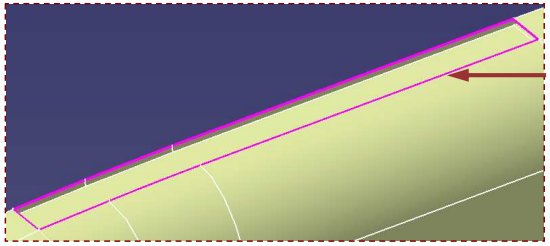
1: Gap (including overlap) greater than Join merging distance



2: Duplicated or embedded surfaces which have not been rejected by the join operator



3: Invalid element (not detected by the Join operator)



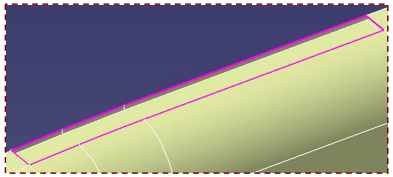
4: Missing element

Student Notes:

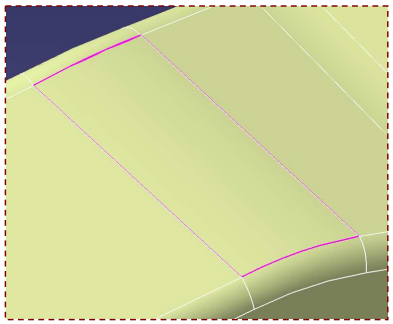
Fixing Free Sides (1/2)

Missing elements, duplications and invalid elements are easy to correct once they are located.

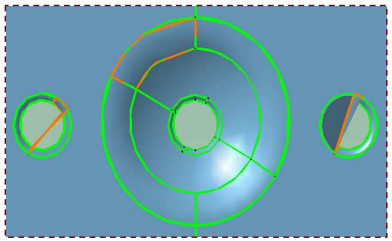
- **Missing element: you can create missing elements by Fill or any other surface creation tool.**



- **Duplicated or embedded element: just remove the unnecessary element from the topology and delete it. As unnecessary copies of duplicated elements are included to the Join as non connex domain, you can also use the Near operator to eliminate all the unwanted non connex domains in one shot.**

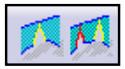


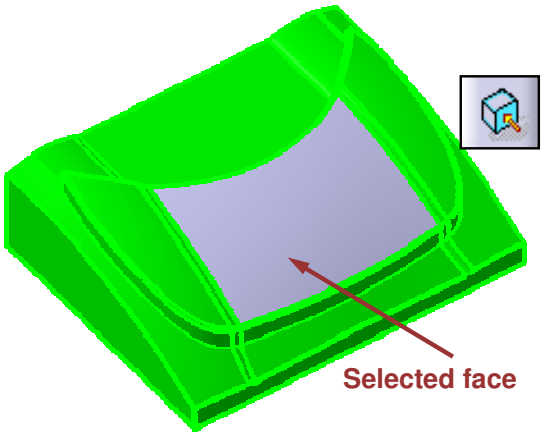
- **Invalid element: remove the invalid element from the topology and recreate it with Fill or any other surface creation tool.**

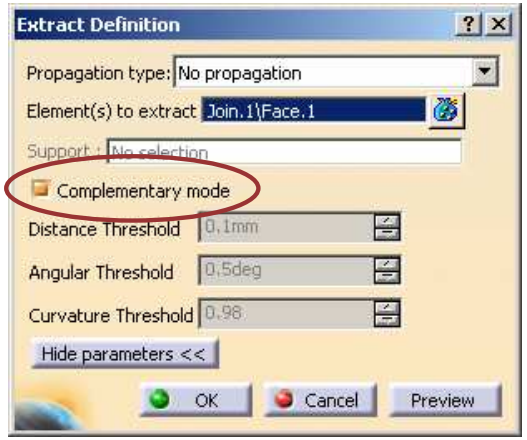


Student Notes:

Fixing Free Sides (2/2)

- Once missing elements, duplications and invalid elements are corrected, only gaps remain
- You can measure gaps with the Surface connection Checker
- Gaps can be corrected:
  - Only at the topological level by increasing the join merging distance with Join or with Local Join.
    Gaps still exist on the geometry, they are only hidden
  - At both topological and geometrical levels:
    - by removing the face which is badly connected to the others and replacing it by a new one. To remove a face from a topology, you can use Extract with the Complementary mode option. The result is a new surface including all faces except the selected one. You can also use Multiple Extract of GSD to remove several faces in one shot (command not available in HA1 workbench).
    - by Healing or Local Healing 





Local Join (1/2)

- Local Join can be used to close a gap in a surface or between surfaces.
- The concept is to release the tolerance locally, only for the selected edges.
- You can also select a boundary to process all gaps in one shot.
- You can close the gap on the topological level only or both on the topological and geometrical levels.

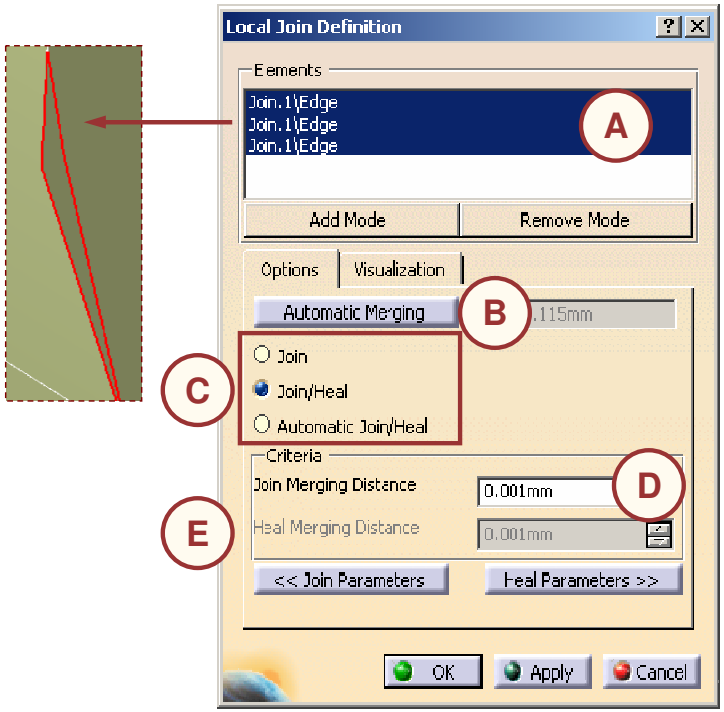
A. List of selected edges or boundary curves.

B. Automatically sets the Join merging distance as equal to the maximum gap size.

C. Choose a simple Join (topological level) or a Join + Healing (topological and geometrical levels). Automatic Join/Heal will do both: join up to the Join tolerance and Heal for bigger gaps.

D. Tolerance for Join.

E. Tolerance for healing when different from join tolerance (in case of Automatic Join/Heal).

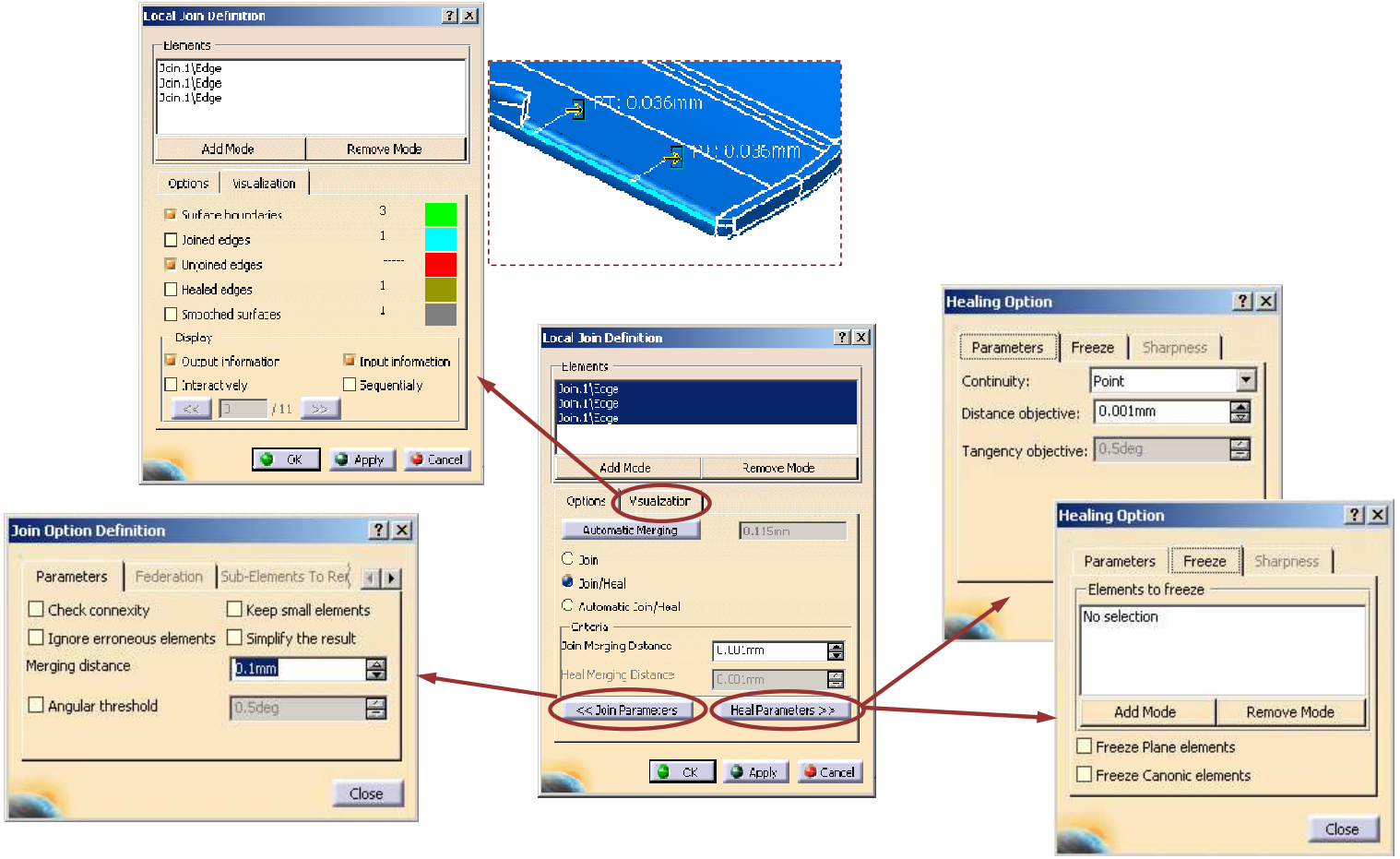


Student Notes:

Local Join (2/2)

Student Notes:

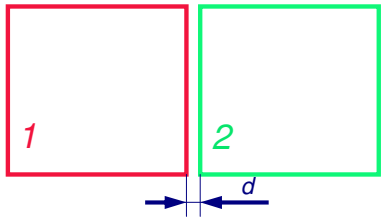
Join and Healing parameters are available to control the result.

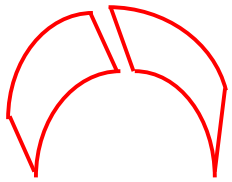


Student Notes:

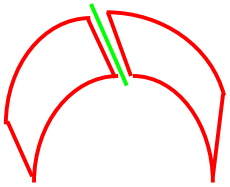
The Healing Operator

- The concept of Healing is to measure connections between faces (same as Join) and to modify the faces when the distance is not within a given tolerance (merging distance).
- The distance is reduced to the user specified Distance Objective.
- Healing = Join + modification of surfaces to close geometry at shared edges.
  - d < 0.001mm (system tolerance): both geometry and topology are closed.
  - 0.001mm < d < merging distance: the geometry is not closed but the topology is closed => HEALING modifies the surfaces to close the geometry.
  - d > merging distance : the geometry is not closed but HEALING does not modify anything because the topology is also not closed.

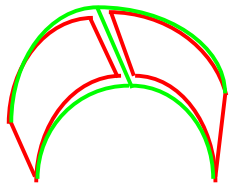




surfaces with a gap



a new common edge is computed



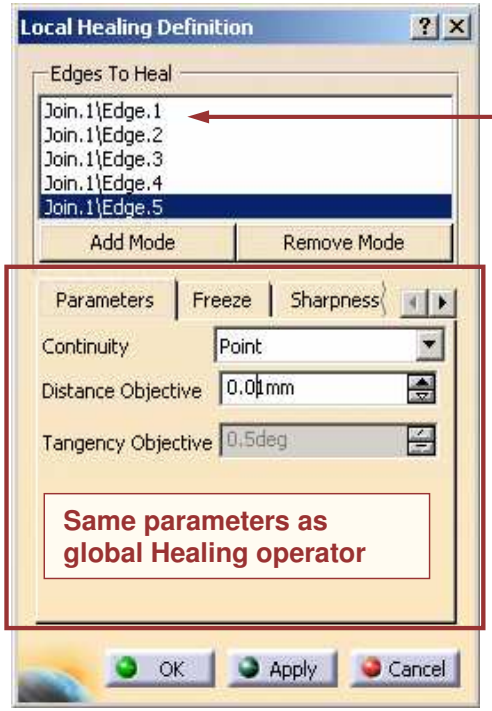
a smart modification is applied to both surfaces

The process is totally automatic: the whole surface is processed.
HEALING fills gaps but can also make surfaces tangent (G1), this can be useful for offset surfaces
The deformation preserves the shape tendency.
Specific faces can be frozen so that they are not deformed (examples: planar faces, canonic shapes).

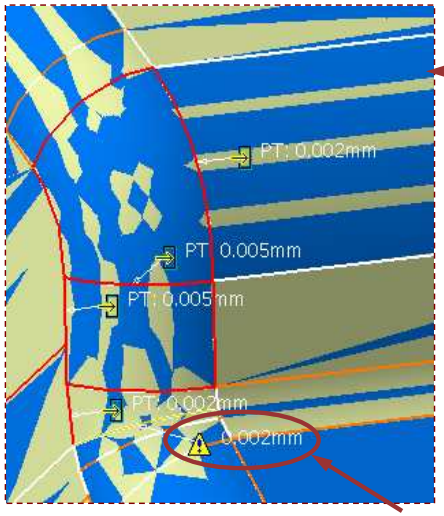
Student Notes:

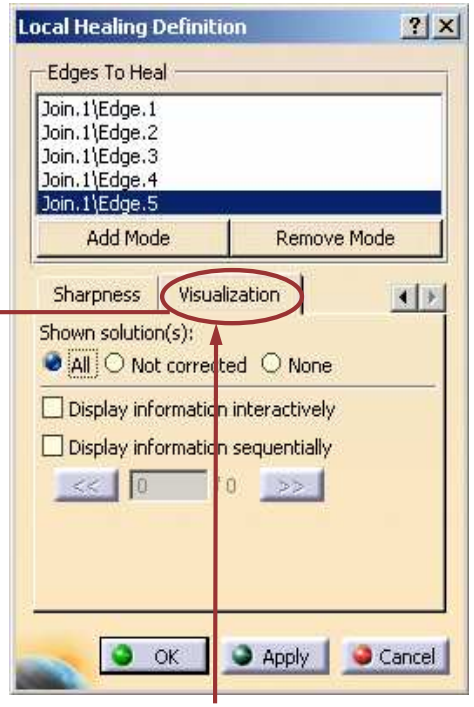
Local Healing

- **Local Healing lets you select shared edges of a surface, and all the faces which share a selected edge are geometrically corrected.**
- **It does not include a Join computation, selected edges must be shared, no healing will be applied to free sides.**



List of selected shared edges (all of them must belong to the same surface).





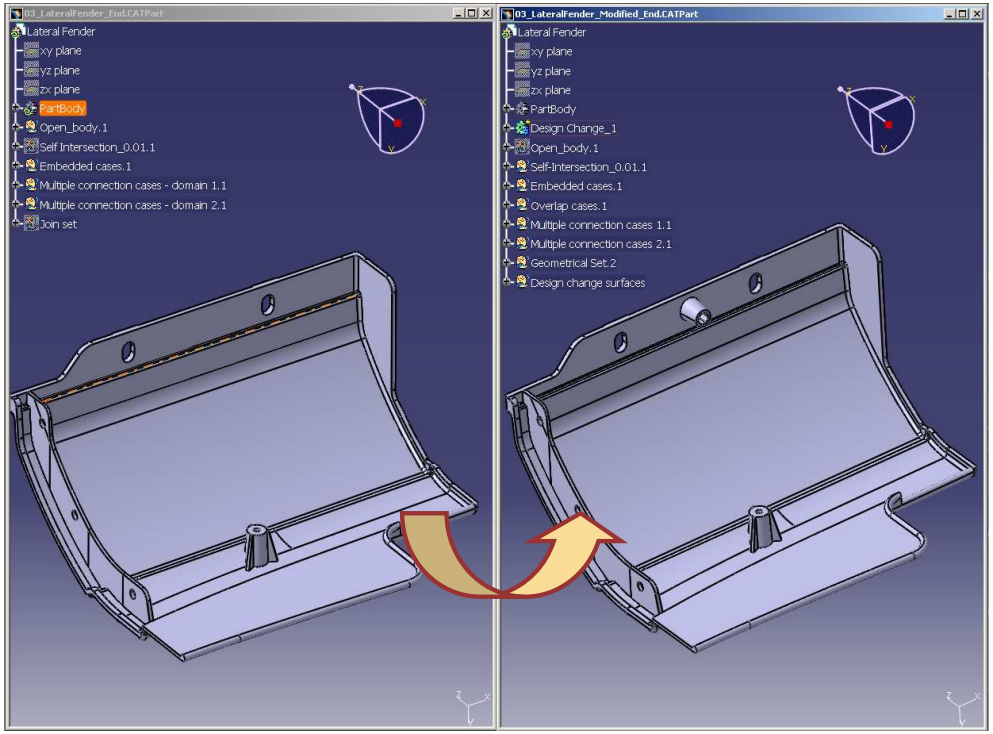
The input gap widths can be displayed, with a warning flag when the objective distance is not reached.

Student Notes:

Comparing Models

In this lesson, you will learn about:

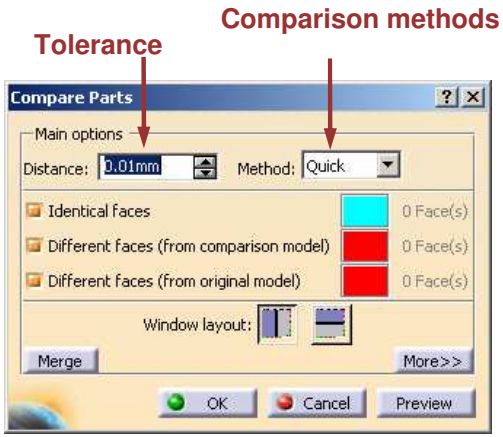
- Comparing two versions of a part.
- Finding differences between them.
- Sorting out the unchanged and the modified areas of the part.



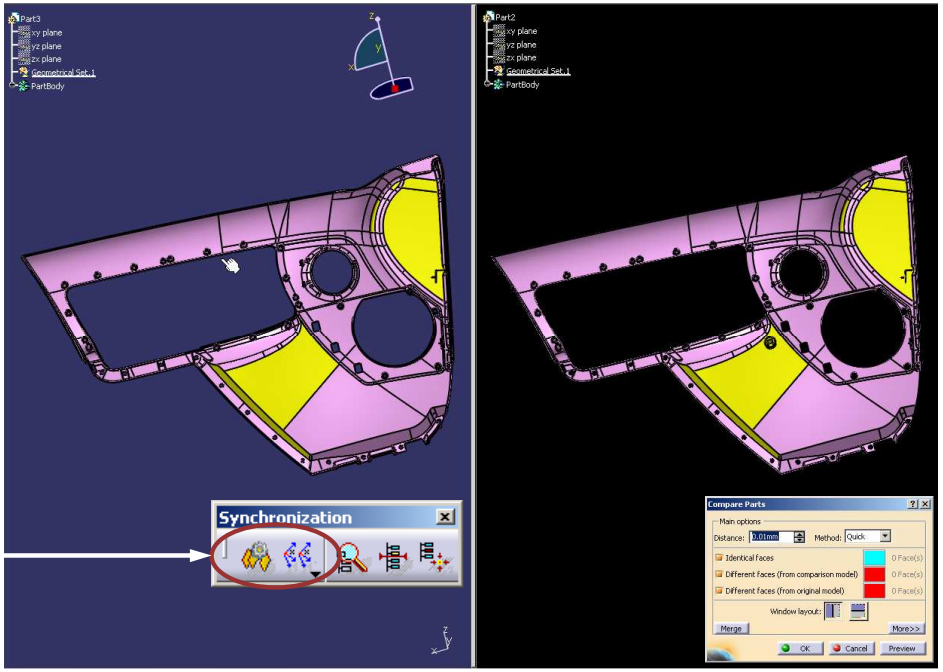
Student Notes:

Compare Parts (1/3)

- Two versions of a same part can be compared with Compare Parts.
- You can easily display both parts in synchronized windows and,
  - find the surfaces or faces which are the same in both parts
  - find the surfaces or faces which are different
  - find embedded surfaces in a part
- The command performs a geometric comparison face to face or surface to surface, it does not compare features or specification trees.



View synchronization options :
- Synchronize commands = apply same command to both views (ex: Hide/Show)
- Synchronize views for zoom and pan

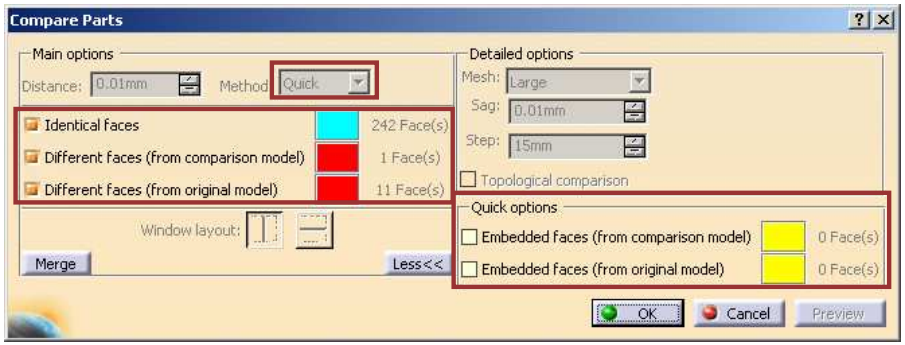


Student Notes:

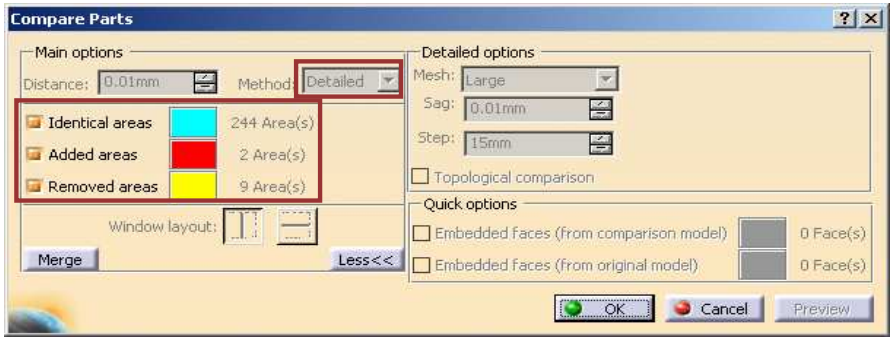
Compare Parts (2/3)

- You can Select the comparison methods like 
  - Quick: It enables a quicker detection of the faces that are different.
  - Detailed: It is a precise comparison.
- You can use the Merge button to sort the surfaces

Quick



Detailed

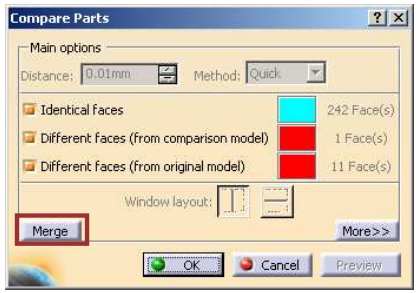


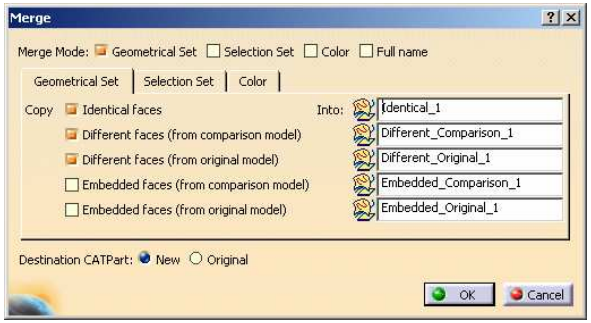
Student Notes:

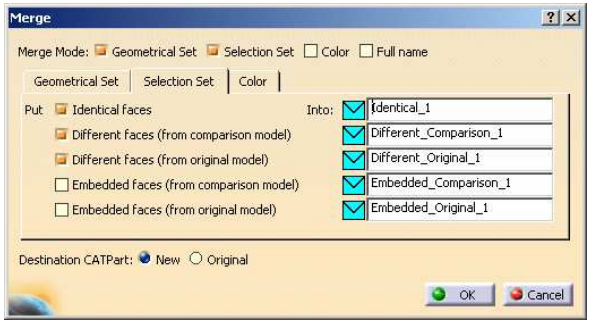
Compare Parts (3/3)

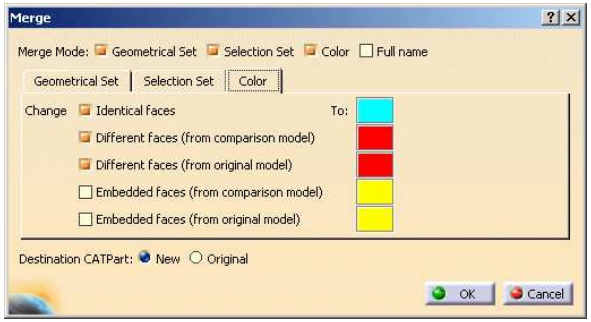
- You can sort the surfaces by

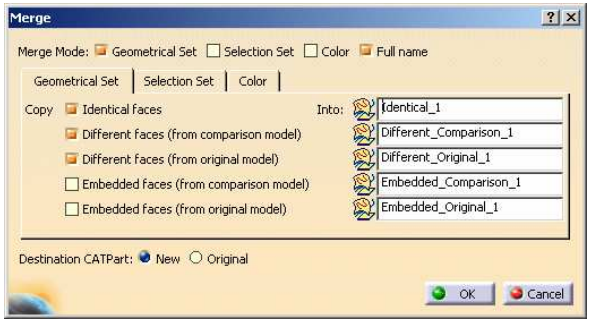
  - Geometrical set: choose names for the geometrical sets to create
  - Selection Set: choose names for the sets to create
  - Color: choose colors
  - Full name: Displays full name of the surfaces in the specification tree
  - Destination: Choose the destination CatPart











Student Notes:

Customizing the Workbench

You will learn about:

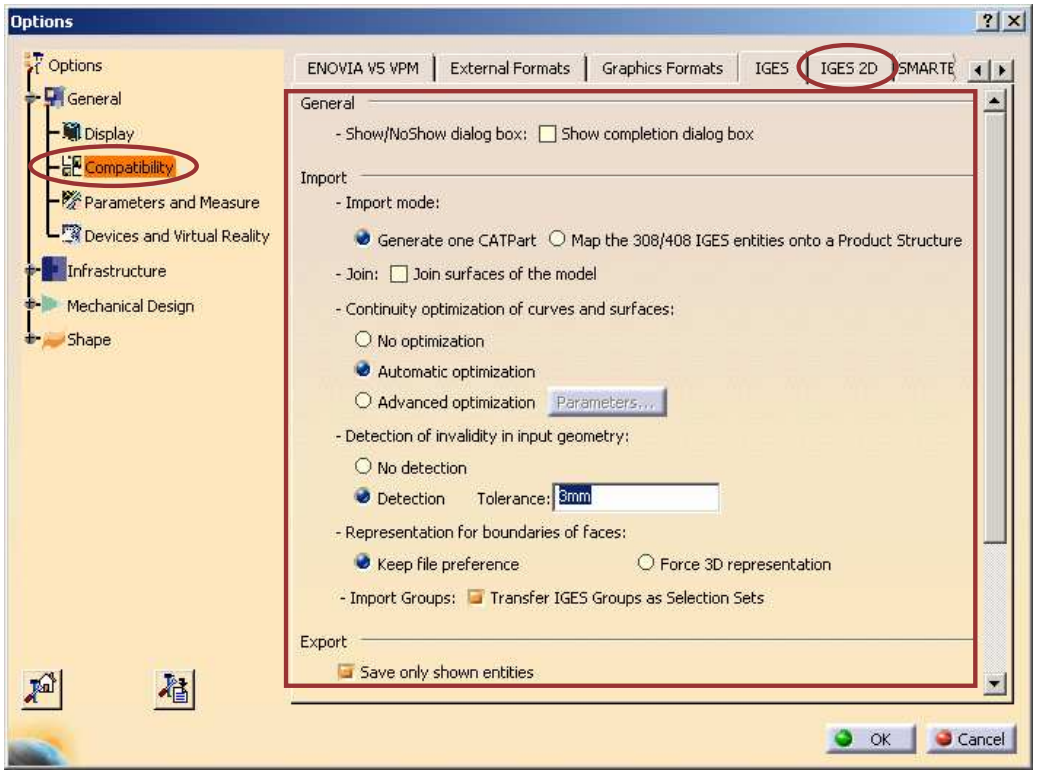
- Controlling and improving the results of IGES 3D import.
- Controlling and improving the results of CATIA V4 import.

IGES 3D Import (1/7)

Student Notes:

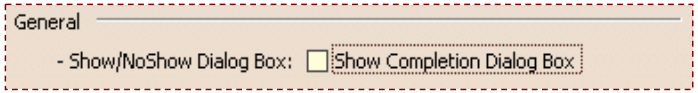
The conversion of an IGES file is started by File/Open.

There are related parameters in the Tools/Options box:

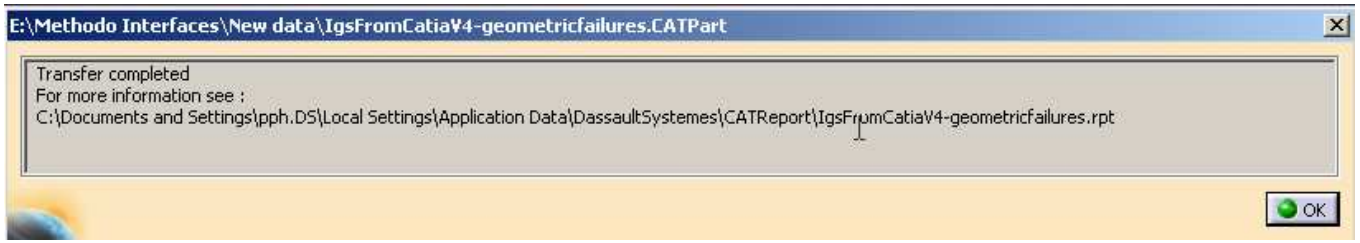


Student Notes:

IGES 3D Import (2/7)



The general parameter can be used to activate or deactivate the display of the completion dialog box at the end of each file import.

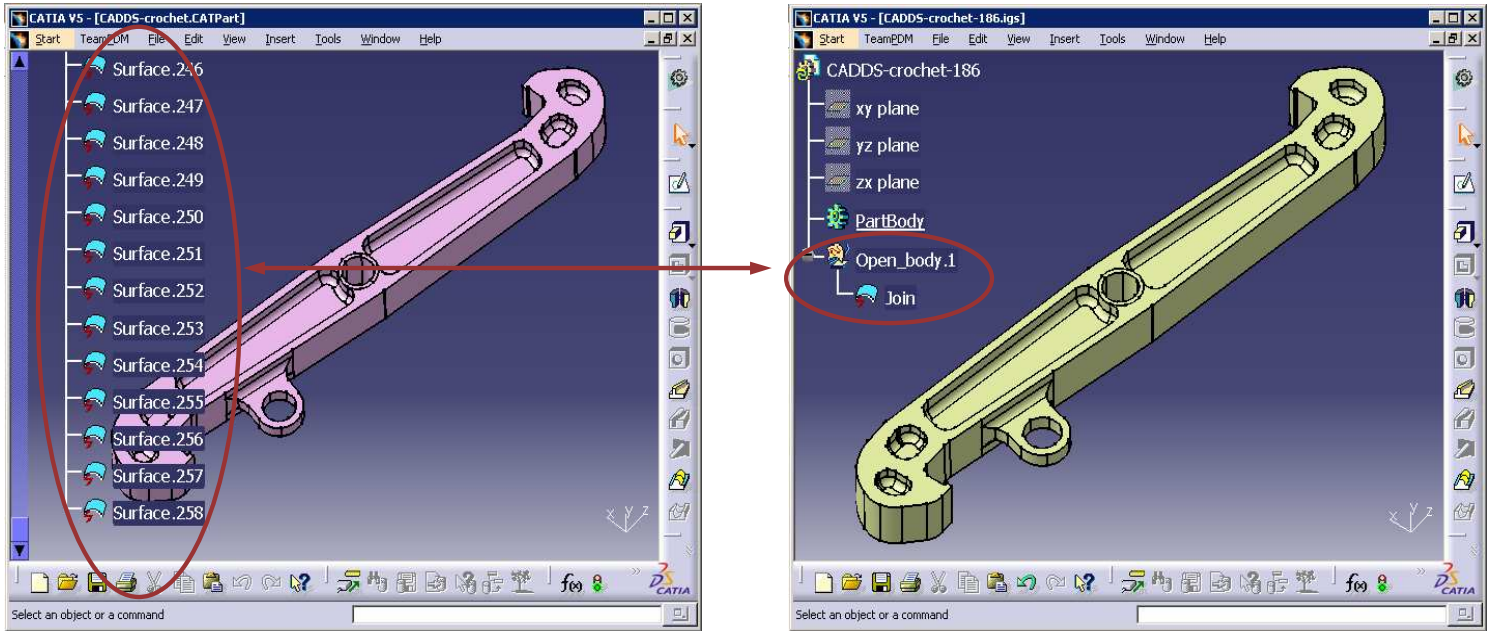


This is specially useful when converting several IGES files together. You do not have to close the message box at the end of the conversion of each file.

Student Notes:

IGES 3D Import (3/7)

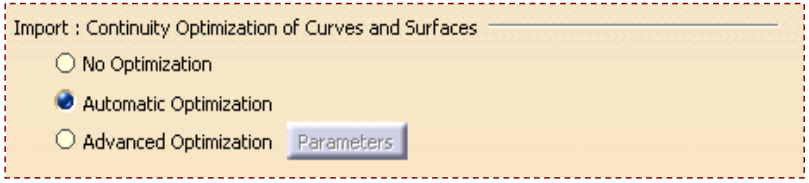
If the Join option is activated, all the surfaces of the IGES file will be joined during the conversion, using the specified tolerance as a merging distance.



Student Notes:

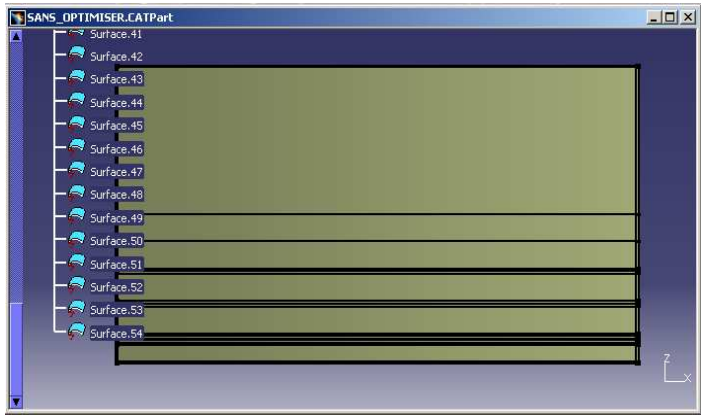
IGES 3D Import (4/7)

The option may be necessary to reduce the amount of data resulting from the conversion.

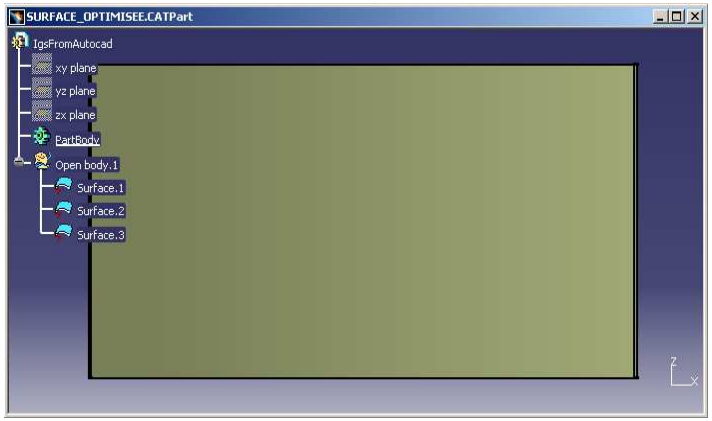


In CATIA V5, a single-cell curve or surface must be C2. If the IGES file contains a curve or surface which is not C2 enough, CATIA will cut it at each C2 discontinuity to create a multi-cell curve or surface. In case of poor quality IGES data, this may cause a drastic increase of the model size.

To prevent this, curves and surfaces of the IGES file can be re-approximated by exactly C2 curves or surfaces, within a specified tolerance, so that they can be converted into single-cell elements.



No C2 optimization: many faces

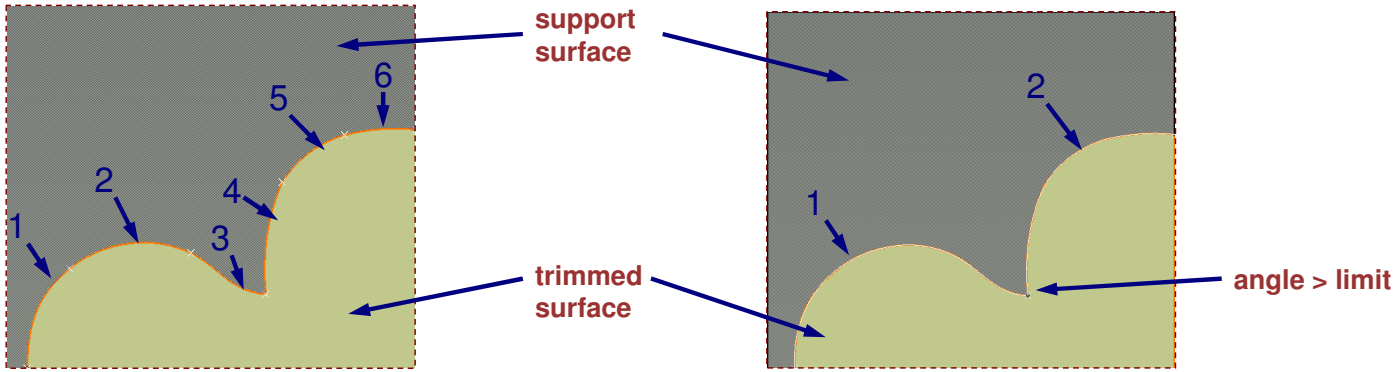


C2 optimization: one single face

Student Notes:

IGES 3D Import (5/7)

Besides, boundaries of IGES surfaces can be made of an unnecessarily high number of curves. The topological reduction option allows to re-approximate the boundaries with a smaller number of curves.

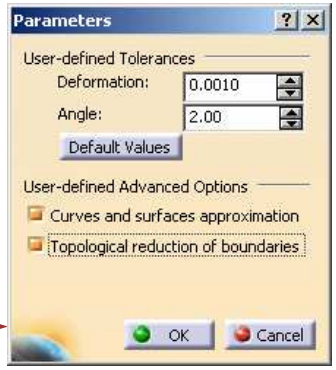


No reduction: 6 curves

Topological reduction: 2 curves

Automatic optimization: the tolerance for the approximation of curves and surfaces is 0.001mm, no topological reduction is done

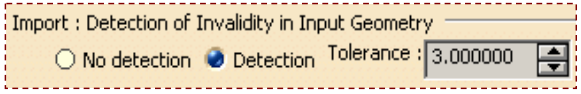
Advanced optimization: you choose the type of data reduction to perform and the parameters to take into account for approximation.



Student Notes:

IGES 3D Import (6/7)

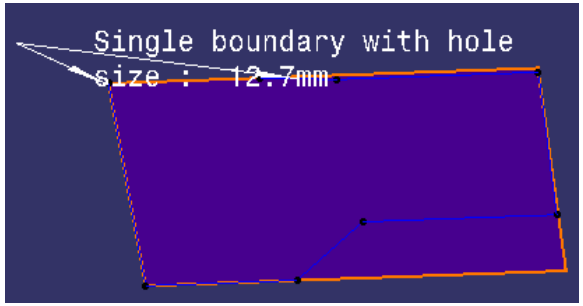
It is also possible to analyze the IGES data to find out the invalid surfaces if any.

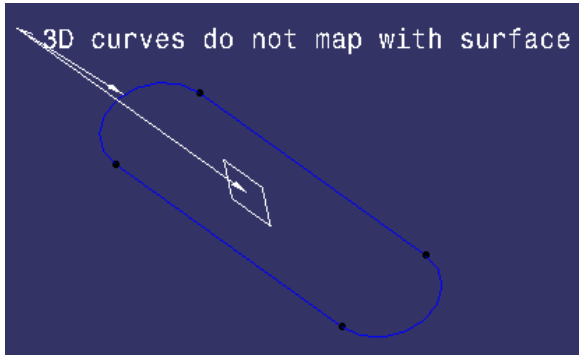


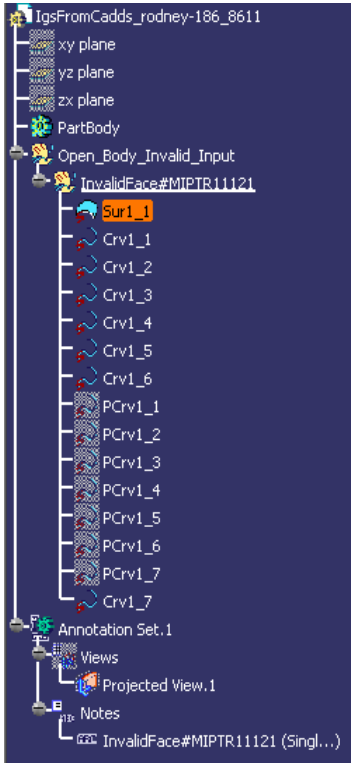
If the option is activated, all surfaces of the IGES file are analyzed to detect:

- **holes in boundary loops: if a hole is wider than the specified tolerance the surface is declared invalid,**
- **boundaries out of support surface: if the curves are further than the specified tolerance from the support surface the element is declared invalid.**

The resulting CATIA part contains an individual geometrical set for each invalid element. These bodies contain the support surface and boundary curves of the element. Annotations are also created to describe the detected invalidities.

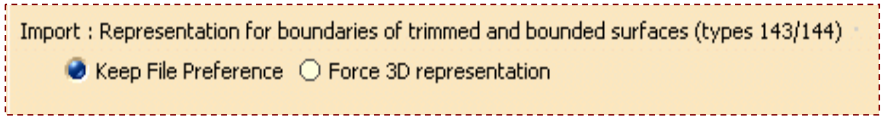






Student Notes:

IGES 3D Import (7/7)

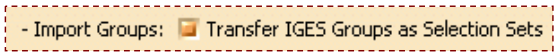


In 3D IGES, trimmed surfaces are described by entity types 143 (Bounded Surface) or 144 (Trimmed Surface). In both cases, the trimmed surface is given by a support surface, 3D curves, 2D curves (P-curves).

The 2D and 3D descriptions of the boundaries are redundant. Normally the emitting system specifies in the header of the IGES file which one is best. This information is not always relevant, and CATIA still lets you choose which one will be used for the conversion:

Option = keep file preference: CATIA will use the type of curves specified in the header (2D or 3D representation),

Option = Force 3D representation: CATIA will use the 3D curves even if the header specifies that 2D curves are preferable; the 2D curves are then computed by projection.



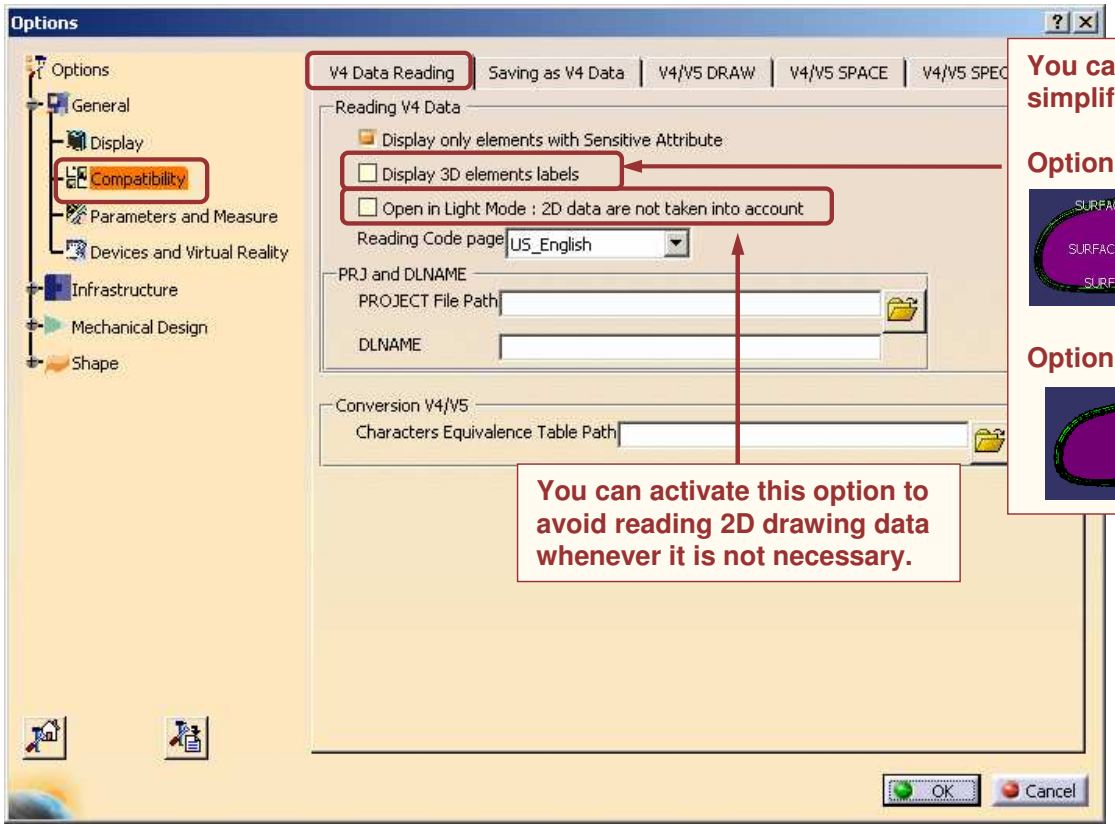
Finally you can choose whether you want to create Selections Sets when importing IGES groups (entity type 402).

Student Notes:

CATIA V4 Import (1/2)

The import of a CATIA V4 model file is started by File/Open.

There are related parameters in the Tools/Options box:



You can activate this option to avoid reading 2D drawing data whenever it is not necessary.

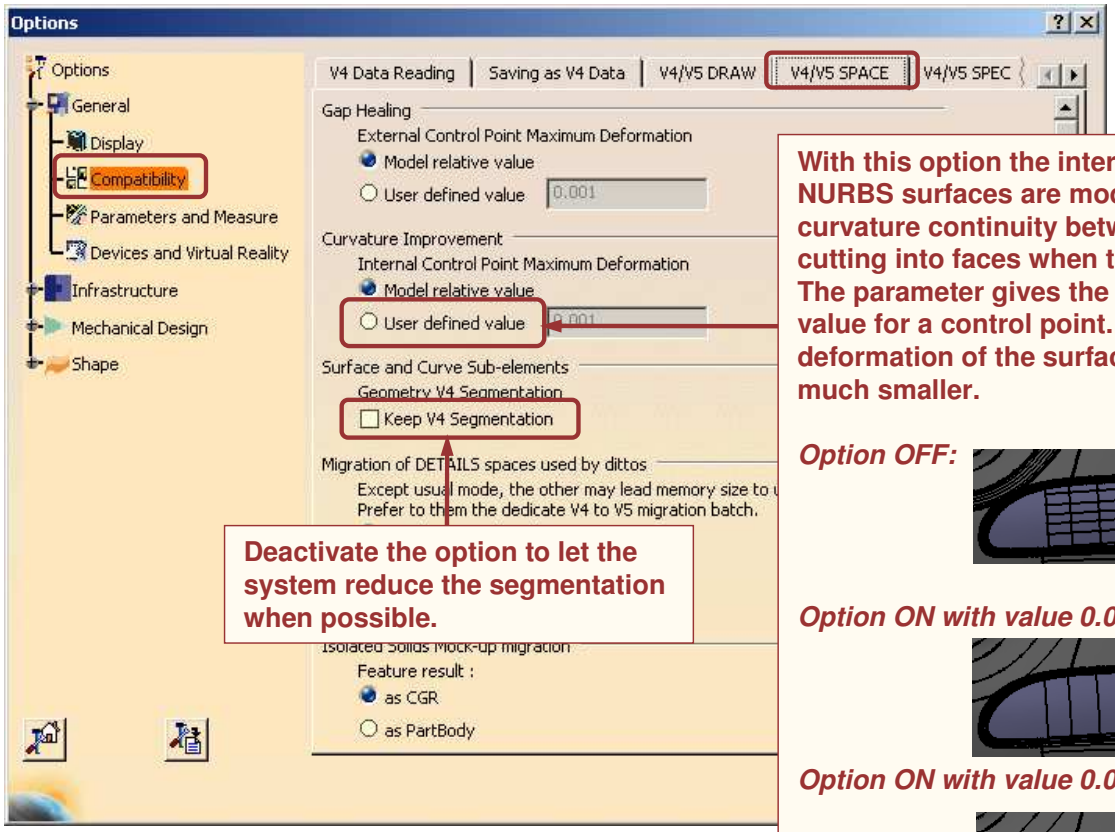
You can deactivate this option to simplify the display:

Option ON:

Option OFF:

Student Notes:

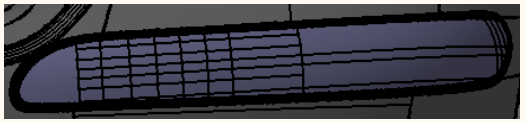
CATIA V4 Import (2/2)



Deactivate the option to let the system reduce the segmentation when possible.

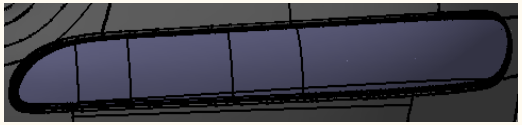
With this option the internal control points of the V4 NURBS surfaces are modified in order to improve the curvature continuity between segments and avoid cutting into faces when the continuity is not C2. The parameter gives the maximum displacement value for a control point. The corresponding deformation of the surface is strictly smaller, usually much smaller.

Option OFF:



Option ON with value 0.005 mm:

Option ON with value 0.01 mm:



To Sum Up

In this course you have seen:

- **How to analyze an imported model**
- **How to repair invalid data**
- **How to repair invalid topological configurations**
- **How to create a valid CATIA V5 topology**
- **How to analyze the free sides of a surface**
- **How to fix free sides topologically and geometrically**
- **How to compare versions of a same part**

Student Notes: