Modeling Algorithms

Shape Healing User's Guide

Version 6.3 / September 2008

NOTICE FOR USERS:

This User Guide is a general instruction for Open CASCADE study. It may be incomplete and even contain occasional mistakes, particularly in examples, samples, etc. Open CASCADE S.A.S. bears no responsibility for such mistakes. If you find any mistakes or imperfections in this document, or if you have suggestions for improving this document, please, contact us and contribute your share to the development of Open CASCADE Technology: bugmaster@opencascade.com

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Contents

1. *Overview*

This manual explains how to use Shape Healing. It provides basic documentation on its operation. For advanced information on Shape Healing and its applications, see our offerings on our web site at *www.opencascade.com/support/training/*

The **Shape Healing** toolkit provides a set of tools to work on the geometry and topology of Open CASCADE shapes. Shape Healing adapts shapes so as to make them as appropriate for use by Open CASCADE as possible.

Shape Healing currently includes several packages that are designed to help you to:

• analyze shape characteristics and, in particular, identify shapes that do not comply with Open CASCADE validity rules

• fix some of the problems shapes may have

• upgrade shape characteristics for users needs, for example a C0 supporting surface can be upgraded so that it becomes C1 continuous.

Each sub-domain has its own scope of functionality:

analysis -	exploring shape properties, computing shape features, detecting violation of Open CASCADE requirements (shape itself is not modified),
fixing -	fixing shape to make it meet the Open CASCADE requirements (the shape may change its original form: modifying, removing, constructing sub-shapes, etc.),
upgrade -	shape improvement for better usability in Open CASCADE or other algorithms (the shape is replaced with a new one, but geometrically they are the same),
customization -	modifying shape representation to fit specific needs (shape is not modified, only the form of its representation is modified),
processing -	mechanism of managing modification of shape via users-editable resource file.

Message management is used for creating messages, filling them with various parameters and storing them in the trace file. This tool provides functionality for attaching messages to the shapes for deferred analysis of various run-time events. In this document only general principles of using ShapeHealing will be described. For more detailed information please see the corresponding CDL files.

Tools responsible for analysis, fixing and upgrading of shapes can give the information about how these operations were performed. This information can be obtained by the user with the help of mechanism of status querying.

1.1. Querying the statuses

Each of fixing and upgrading tools has its own status, which is reset when their methods are called. The status can contain several flags, which give the information about how the method was performed. For exploring the statuses, a set of methods named Status...() is provided. These methods accept enumeration ShapeExtend_Status and return True if the status has the corresponding flag set. The meaning of flags for each method is described below.

The status may contain a set of Boolean flags (internally represented by bits). Flags are coded by enumeration ShapeExtend_Status. This enumeration provides the following families of statuses:

ShapeExtend_OK -	the situation is OK, no operation is necessary and has not been performed,
ShapeExtend_DONE -	the operation has been successfully performed,
ShapeExtend_FAIL -	an error has occurred during operation.

The user can test the status for the presence of some flag(s), using Status...() method(s) provided by the class:

```
    if ( object.Status.. ( ShapeExtend_DONE ) ) {// something was done
    }
```

8 'DONE' and 8 'FAIL' flags, named ShapeExtend_DONE1 ... ShapeExtend_FAIL8, are defined for a detailed analysis of the situation encountered. Each method assigns its own meaning to each flag, documented in the CDL for that method. There are also three enumerative values used for testing several flags at a time:

ShapeExtend_OK -	if no flags have been set,
ShapeExtend_DONE -	if at least one ShapeExtend_DONEi has been set,
ShapeExtend_FAIL -	if at least one ShapeExtend_FAILi has been set,

2. *Repair*

Algorithms for fixing problematic (violating the Open CASCADE requirements) shapes are placed in package ShapeFix.

Each class of package ShapeFix deals with one certain type of shapes or with some family of problems.

There is no necessity for you to detect problems before using ShapeFix because all components of package ShapeFix make an analysis of existing problems before fixing them by a corresponding tool from package of ShapeAnalysis and then fix the discovered problems.

The ShapeFix package currently includes functions that:

- add a 2D curve or a 3D curve where one is missing,
- correct a deviation of a 2D curve from a 3D curve when it exceeds a given tolerance value,
- limit the tolerance value of shapes within a given range,
- set a given tolerance value for shapes,
- repair the connections between adjacent edges of a wire,
- correct self–intersecting wires,
- add seam edges,
- correct gaps between 3D and 2D curves,
- merge and remove small edges,
- correct orientation of shells and solids.

2.1. The easiest solution.

.

The simplest way for fixing shapes is to use classes ShapeFix_Shape and ShapeFix_Wireframe on a whole shape with default parameters. A combination of these tools can fix most of problems that shapes may have.

For fixing a shape you can use the following sequence of actions:

Create the tool ShapeFix_Shape and initialize it by shape.

```
Handle(ShapeFix_Shape) sfs = new ShapeFix_Shape;
sfs->Init ( shape );
```

Set the basic precision and the maximum allowed tolerance:

```
sfs->SetPrecision ( Prec );
sfs->SetMaxTolerance ( maxTol );
```

Where Prec – basic precision, maxTol – maximum allowed tolerance.

Please NOTE:

- *All problems will be detected for cases when a dimension of invalidity is larger than the basic precision or a tolerance of sub-shape on that problem is detected.*

- *The maximum tolerance value limits the increasing tolerance for fixing a problem. If a value larger than the maximum allowed tolerance is necessary for correcting a detected problem the problem can not be fixed.*

Make fixing

```
sfs->Perform();
```

Get result

```
TopoDS_Shape aResult = sfs->Shape();
```

But in some cases using one tool only (ShapeFix_Shape) can be insufficient. It is possible to use tools for merging and removing small edges and fixing gaps between 2D and 3D curves:

Create the tool ShapeFix_Wireframe and initialize it by shape.

```
Handle(ShapeFix_Wirefarme) SFWF = new ShapeFix_Wirefarme(shape);
```

Or

```
Handle(ShapeFix_Wirefarme) SFWF = new ShapeFix_Wirefarme;
SFWF->Load(shape);
```

Set the basic precision and the maximum allowed tolerance:

```
sfs->SetPrecision ( Prec );
sfs->SetMaxTolerance ( maxTol );
```

Please see the description for Prec and maxTol above.

Merging and removing small edges

```
//set mode for removing small edges
SFWF->DropSmallEdgesMode() = Standard_True;
SFWF->FixSmallEdges();
```

Please NOTE:

Small edges are not removed with the default mode. But in many cases removing small edges is very useful for fixing a shape.

Fixing gaps for 2D and 3D curves

```
SFWF->FixWireGaps();
```

Getting the result

```
TopoDS_Shape Result = SFWF-Shape();
```

2.2. How you manage to correct shapes.

If you do not want to make fixes on the whole shape or make a definite set of fixes you can set flags for separate fix cases (marking them ON or OFF) and you can also use classes for fixing specific types of sub-shapes such as solids, shells, faces, wires, etc…

For each type of sub-shapes there are specific types of fixing tools such as ShapeFix_Solid, ShapeFix_Shell, ShapeFix_Face, ShapeFix_Wire etc…

2.2.1. Fixing sub-shapes.

If you want to make a fix on one subshape of a certain shape it is possible to take the following steps:

- create a tool for a specified subshape type and initialize this tool by the subshape
- create a tool for rebuilding the shape and initialize it by the whole shape (section 5.1)
- set a tool for rebuilding the shape in the tool for fixing the subshape
- fix the subshape
- get the resulting whole shape containing a new corrected subshape

Example

Fixing one face Face1 of shape Shape1

```
//creates tools for fixing a face
Handle(ShapeFix_Face)  SFF= new ShapeFix_Face;

// creates tool for rebuilding a shape and initializes it by shape
Handle(ShapeBuild_ReShape) Context = new ShapeBuild_ReShape;
Context->Apply(Shape1);

//set a tool for rebuilding a shape in the tool for fixing
SFF->SetContext(Context);

//initialize the fixing tool by one face
SFF->Init(Face1);

//fixing the set face
SFF->Perform();

//getting the result
TopoDS_Shape  NewShape = Context->Apply(Shape1);
//Resulting shape contains the fixed face.
```

A set of required fixes and invalid sub-shapes can be obtained with the help of tools responsible for the analysis of shape validity (section 3.2).

2.3. Short description of repairing tools

Each class of package ShapeFix deals with one certain type of shapes or with a family of problems. Each of repairing tools makes fixes for a specified shape and its sub-shapes with the help of method Perform() containing an optimal set of fixes. The execution of these fixes in the method Perform can be managed with the help of a set of control flags (fixes can be either forced or forbidden).

2.3.1. General principles of repairing tools use.

To perform fixes the following sequence of actions should be applied:

1. create a tool.
2. set the following values:
   - set the working precision by method *SetPrecision()* (default 1.e-7)
   - set the maximum allowed tolerance by method *SetMaxTolerance()* (by default it is equal to the working precision).
   - set the minimum tolerance by method *SetMinTolerance()* (by default it is equal to the working precision).
   - set a tool for rebuilding shapes after the modification (tool ShapeBuild_ReShape) by method *SetContext(*). For separate faces, wires and edges this tool is set optionally.
   - if you want to force or forbid some of fixes it is necessary to set a corresponding flag to 0 or 1.
3. initialize the tool by the shape with the help of methods Init or Load (if it exists)
4. use method *Perform()* or create your own set of fixes.
5. Check the statuses of fixes by the general method *Status* or specialized methods *Status…* (for example *StatusSelfIntersection* (ShapeExtentd_DONE)). See the description of statuses below in the section().
6. get the result. Result will be obtained by two ways :
   - get the result with the help of a special method *Shape(),Face(),Wire().Edge(*).
   - get the result from the rebuilding tool by method *Apply* (for access to rebuilding tool use method *Context(*)):

```
TopoDS_Shape resultShape = fixtool->Context()->Apply(initialShape);
```

Please NOTE.

History of modification of a shape and its sub-shapes can be obtained from the tool for shape re-building (ShapeBuild_ReShape).

```
TopoDS_Shape modifsubshape = fixtool->Context()
                                             ->Apply(initsubshape);
```

The following tools are used for repairing shapes:

ShapeFix_Edge,

ShapeFix_Wire,

ShapeFix_Face,

ShapeFix_Shell,

ShapeFix_Solid,

ShapeFix_Shape,

ShapeFix_Wireframe,

ShapeFix_FixSmallFace,

ShapeFix_ShapeTolerance

The API package method is the following:

ShapeFix::SameParameter().

2.3.2. Managing the flags

Flags named Fix...Mode() are used to control the execution of fixing procedures from the API fixing methods. By default, these flags have values equal to -1, this means that a corresponding procedure will either be called or not called, depending on the situation. If the flag is set to 1, the procedure is executed anyway; if the flag is 0, the procedure is not executed. The name of the flag corresponds to the fixing procedure that is controlled. For each fixing tool there exists its own set of flags. In order to set a flag to the desired value it is necessary to get a tool containing this flag and to set the flag to the required value.

Example

Interdiction of performing fixes for removing small edges - FixSmall

```
Handle(ShapeFix_Shape) Sfs = new ShapeFix_Shape(shape);
Sfs-> FixWireTool ()->FixSmallMode () =0;
if(Sfs->Perform())
  TopoDS_Shape resShape = Sfs->Shape();
```

2.3.3. Repairing tool for shapes (Class ShapeFix_Shape).

This tool fixes shapes by using repairing tools for all sub-shapes of a shape. It provides access to all repairing tools for fixing sub-shapes of specified shape and to all control flags from these tools.

Repairing tools used for fixing sub-shapes of a specified shape.

ShapeFix_Solid.

ShapeFix_Shell,

ShapeFix_Face,

ShapeFix_Wire,

ShapeFix_Edge.

Please NOTE:

By default, the repairing tool will fix all sub-shapes of a specified shape.

This tool gives the ability to fix any type of shapes (compound, solid, shell, face, edge) and to manage fixes by flags.

Example:

Forcing the removal of invalid 2D curves from face.

```
TopoDS_Face face … // face with invalid 2D curves.
//creation of tool and its initialization by shape.
Handle(ShapeFix_Shape) sfs = new ShapeFix_Shape(face);
```

```
//set work precision and max allowed tolerance.
sfs->SetPrecision(prec);
sfs->SetMaxTolerance(maxTol);
//set the value of flag for forcing the removal of 2D curves
sfs->FixWireTool()->FixRemovePCurveMode() =1;
//reform fixes
sfs->Perform();
//getting the result
if(sfs->Status(ShapeExtend_DONE) ) {
 cout<<”Shape was fixed”<<endl;
 TopoDS_Shape resFace = sfs->Shape();
}
else if(sfs->Status(ShapeExtend_FAIL)) {
cout<<” Shape could not be fixed”<<endl;
}
else if(sfs->Status(ShapeExtent_OK)) {
cout<<” Initial face is valid with specified precision
=”<<prec<<endl;
}
```

2.3.4. Repairing tool for solids(Class ShapeFix_Solid)

This tool is intended for the fixing of solids and building a solid from a shell to obtain a valid solid with a finite volume. For correction of shells belonging to solid tool ShapeFix_Shell is used.

Control flags

FixShellMode - Mode for applying fixes of ShapeFix_Shell, True by default.

CreateOpenShellMode - If it is equal to true solids are created from open shells, else solids are created from closed shells only, False by default.

2.3.5. Repairing tool for shells (Class ShapeFix_Shell)

This tool is intended to fix the wrong orientation of faces in a shell. It changes orientation of faces in the shell if necessary so that all faces in the shell have coherent orientations. If it is impossible to orient all faces in the shell (like in case of Mebious tape), then a few manifold or non-manifold shells will be created depending on the specified Non-manifold mode. To correct faces in the shell the ShapeFix_Face tool is used.

Control flags

FixFaceMode - mode for applying the fixes of ShapeFix_Face, True by default.

FixOrientationMode - mode for applying a fix for the orientation of faces in the shell.

2.3.6. Repairing tool for faces (Class ShapeFix_Face)

This tool is intended to fix problems on a face with regard to its wires. It allows controlling the creation of a face (adding the wires), and fixing its wires by means of tool ShapeFix_Wire.

When a wire is added to a face, it can be reordered and degenerated edges can be fixed. This is performed or not depending on the user-defined flags (by default, False).

The following fixes are available:

- fixing of orientation of the wires on the face. If the face has no wire, the natural bounds are computed. If the face is on a spherical surface and has two or more wires on it describing holes, the natural bounds are added. In case of a single wire, it is made to be an outer one. If the face has several wires, they are oriented to lay one outside another (if possible). If the supporting surface is periodic, 2D curves of internal wires can be shifted on integer number of periods to put them inside the outer wire.

- fixing the case when the face on the closed surface is defined by a set of closed wires, and the seam is missing (this is not valid in Open CASCADE). In that case, these wires are connected by means of seam edges into the same wire.

Control flags

FixWireMode -mode for applying fixes of a wire, True by default.

FixOrientationMode - mode for orienting a wire to border a limited square, True by default.

FixAddNaturalBoundMode - mode for adding natural bounds to a face, False by default.

FixMissingSeamMode – mode to fix a missing seam, True by default. If True, tries to insert a seam.

FixSmallAreaWireMode - mode to fix a small-area wire, False by default. If True, drops wires bounding small areas.

Example

```
TopoDS_Face face = ...;
TopoDS_Wire wire = ...;

//Creates a tool and adds a wire to the face
ShapeFix_Face sff (face);
sff.Add (wire);

//use method Perform to fix the wire and the face
sff.Perfom();

//or make a separate fix for the orientation of wire on the face
sff.FixOrientation();

//Getting the resulting face
TopoDS_Face newface = sff.Face();
```

2.3.7. Repairing tool for wires (Class ShapeFix_Wire)

This class provides a set of fixes for fixing a wire.

This method Perform() performs all the available fixes in addition to the geometrical filling of gaps. The geometrical filling of gaps can be made with the help of the tool for fixing the wireframe of shape *ShapeFix_Wireframe*.

The fixing order and the default behavior of *Perform()* is as follows:

- Edges in the wire *FixReorder* are reordered; in case it is forbidden, the analysis of whether the wire is ordered or not is performed anyway (this information is used for determining the default behavior of other methods).
- Small edges *FixSmall* are removed.
- Edges in the wire are connected (topologically) *FixConnected* (if the wire is ordered).

- Edges (3Dcurves and 2D curves) *FixEdgeCurves* (without *FixShifted* if the wire is not ordered) are fixed.
- Degenerated edges *FixDegenerated* are added (if the wire is ordered).
- Self-intersection *FixSelfIntersection* is fixed (if the wire is ordered and *ClosedMode* is True).
- Lacking edges *FixLacking* are fixed (if the wire is ordered).

Please NOTE:

Most of fixing methods expect edges in a wire to be ordered, so it is necessary to make call to FixReorder() before making any other fixes.

Some fixes can be made in three ways:

- Increasing the tolerance of an edge or a vertex.
- Changing topology (adding/removing/replacing an edge in the wire and/or replacing the vertex in the edge, copying the edge etc.).
- Changing geometry (shifting a vertex or adjusting ends of an edge curve to vertices, or re-computing a 3D curve or 2D curves of the edge).

When it is possible to make a fix in more than one way (e.g., either by increasing the tolerance or shifting a vertex), it is chosen according to the user-defined flags:

ModifyTopologyMode - allows modifying topology, False by default.

ModifyGeometryMode - allows modifying geometry. Now this flag is used only in fixing self-intersecting edges (allows to modify 2D curves) and is True by default.

The methods of this class correct the following problems. They:

- fix disordered edges in the wire (reorder),
- fix small edges (remove edges with a length less than the given value),
- fix disconnected edges (adjacent edges having different vertices),
- fix the consistency of edge curves,
- fix degenerated edges,
- fix intersections of 2D curves of the edges,
- fix lacking edges to fill gaps in the parametrical space of a surface,
- fix gaps in 2D and 3D wires by means of geometrical filling.

Fixing disordered edges

This fix is necessary for most of other fixes (but is not necessary for Open CASCADE). It checks whether edges in the wire go in a sequential order (the end of a preceding edge is the start of a following one). If it is not so, an attempt to reorder the edges is made.

Fixing small edges

This fixing method searches for the edges, which have a length less than the given value (degenerated edges are ignored). If such an edge is found, it is removed provided that one of the following conditions is satisfied:

- both end vertices of that edge are one and the same vertex,
- end vertices of the edge are different, but the flag *ModifyTopologyMode* is True. In the latter case, method *FixConnected* is applied to the preceding and the following edges to ensure their connection.

Fixing disconnected edges

This method forces two adjacent edges to share the same common vertex (if they do not have a common one). It checks whether the end vertex of the preceding edge coincides with the start vertex of the following edge with the given precision, and then creates a new vertex and sets it as a common vertex for the fixed edges. At that point, edges are copied, hence the wire topology is changed (regardless of the *ModifyTopologyMode* flag). If the vertices do not coincide, this method fails.

Fixing the consistency of edge curves

This method performs a set of fixes dealing with 3D curves and 2D curves of edges in a wire.

These fixes will be activated with the help of a set of fixes from the repairing tool for edges called *ShapeFix_Edge*. Each of these fixes can be forced or forbidden by means of setting the corresponding flag to either True or False. The mentioned fixes and the conditions of their execution are:

- fixing a disoriented 2D curve by call to *ShapeFix_Edge::FixReversed2d*- if not forbidden,
- removing a wrong 2D curve by call to *ShapeFix_Edge::FixRemovePCurve* - only if forced,
- fixing a missing 2D curve by call to *ShapeFix_Edge::FixAddPCurve* - if not forbidden,
- removing a wrong 3D curve by call to *ShapeFix_Edge::FixRemoveCurve3d*- only if forced,
- fixing a missing 3D curve by call to *ShapeFix_Edge::FixAddCurve3d* - if not forbidden,
- fixing 2D curves of seam edges - if not forbidden.
- fixing 2D curves which can be shifted at an integer number of periods on the closed surface - if not forbidden. This fix is required if 2D curves of some edges in a wire lying on a closed surface were recomputed from 3D curves. In that case, the 2D curve for the edge, which goes along the seam of the surface, can be incorrectly shifted at an integer number of periods. The method detects such cases and shifts wrong 2D curves back, ensuring that the 2D curves of the edges in the wire are connected,
- fixing the SameParameter problem by call to *ShapeFix_Edge::FixSameParameter* - if not forbidden.

Fixing degenerated edges

This method checks whether some edge in a wire lies on a degenerated point of the supporting surface, or whether there is a degenerated point between the edges. If one of these cases is detected for some edge, a new degenerated edge is created and it replaces the current edge in the first case or is added to the wire in the second case. The newly created degenerated edge has a straight 2D curve, which goes from the end of the 2D curve of the preceding edge to the start of the following one.

Fixing intersections of 2D curves of the edges

This method detects and fixes the following problems:

- self-intersection of 2D curves of individual edges. If the flag *ModifyGeometryMode()* is False this fix will be performed by increasing the tolerance of one of end vertices to a value less then *MaxTolerance()*.
- intersection of 2D curves of each of the two adjacent edges (except the first and the last edges if the flag ClosedWireMode is False). If such intersection is found, the common vertex is modified in order to comprise the intersection point. If the flag *ModifyTopologyMode* is False this fix will be performed by increasing the tolerance of the vertex to a value less then *MaxTolerance()*.
- intersection of 2D curves of non-adjacent edges. If such intersection is found the tolerance of the nearest vertex is increased to comprise the intersection point. If such increase cannot be done with a tolerance less than MaxTolerance this fix will not be performed.

Fixing a lacking edge

This method checks whether a wire is not closed in the parametrical space of the surface (while it can be closed in 3D). This is done by checking whether the gap between 2D curves of each of the two adjacent edges in the wire is smaller than the tolerance of the corresponding vertex. The algorithm computes the gap between the edges, analyses positional relationship of the ends of these edges and (if possible) tries to insert a new edge into the gap or increases the tolerance.

Fixing gaps in 2D and 3D wire by geometrical filling

These methods check gaps between the ends of 2D or 3D curves of adjacent edges. Boolean flag *FixGapsByRanges* is used to activate an additional mode applied before converting to B-Splines. When this mode is on, methods try to find the most precise intersection of curves, or the most precise projection of a target point, or an extremity point between two curves (to modify their parametric range accordingly). This mode is off by default. Independently of the additional mode described above, if gaps remain, these methods convert curves to B-Spline form and shift their ends if a gap is detected.

Method FixGap2d moves the ends of 2D curves to the middle point. Method FixGaps3d moves the ends of 3D curves to a common vertex.

Control flags

ClosedWireMode -specifies whether the wire is (or should be) closed or not. If that flag is True (by default), fixes that require or force connection between edges are executed for the last and the first edges also, otherwise they are not.

FixReorderMode,

FixSmallMode,

FixConnectedMode,

FixEdgeCurvesMode,

FixDegeneratedMode,

FixSelfIntersectionMode,

FixLackingMode.

The following flags are defined for method FixEdgeCurves():

FixReversed2dMode,

FixRemovePCurveMode,

FixRemoveCurve3dMode,

FixAddPCurveMode,

FixAddCurve3dMode,

FixSeamMode,

FixShiftedMode,

FixSameParameterMode.

The following flags are defined for method FixSelfIntersection():

FixSelfIntersectingEdgeMode,

FixIntersectingEdgesMode.

Example:

Creation of your own set of fixes.

```
    TopoDS_Face face = ...;
    TopoDS_Wire wire = ...;
    Standard_Real precision = 1e-04;
```

```
ShapeFix_Wire sfw (wire, face, precision);
//Creates a tool and loads objects into it
sfw.FixReorder();
//Orders edges in the wire so that each edge
//starts at the end of the one before it
sfw.FixConnected();
//Forces all adjacent edges to share
//the same vertex
Standard_Boolean LockVertex = Standard_True;
if (sfw.FixSmall (LockVertex, precision)) {
  //Removes all edges which are shorter than
  //the given precision and have the same vertex at both ends
}
if (sfw.FixSelfIntersection()) {
  //Fixes self-intersecting edges and intersecting
  //adjacent edges
  cout<<"Wire was slightly self-intersecting. Repaired"<<endl;
}
if ( sfw.FixLacking ( Standard_False ) ) {
  //Inserts edges to connect adjacent
  //non-continuous edges
}
TopoDS_Wire newwire = sfw.Wire();
//Returns the corrected wire
```

2.3.8. Repairing tool for edges (Class ShapeFix_Edge)

This class provides tools for fixing invalid edges. The following geometrical and/or topological inconsistencies are detected and fixed:

- missing 3D curve or 2D curve,
- mismatching orientation of a 3D curve and a 2D curve,
- incorrect SameParameter flag (curve deviation is greater than the edge tolerance).

Each fixing method first checks whether the problem exists using methods of the *ShapeAnalysis_Edge* class. If the problem is not detected, nothing is done.

Please NOTE:

This tool does not have the method Perform().

Example:

```
TopoDS_Face face = ...;
TopoDS_Wire wire = ...;
Standard_Real precision = 1e-04;
ShapeFix_Edge sfe;
```

```
//Creates a tool to work on edges
for(TopExp_Explorer wireexp(wire,TopAbs_EDGE);wireexp.More();
                   wireexp.Next()) {
  TopoDS_Edge edge = TopoDS::Edge(wireexp.Current());
  if (sfe.FixAddPCurve (edge, face, Standard_False,
                   BRep_Tool::Tolerance (edge))) {
    //Creates and adds pcurve to the
    //edge on the face
  }
  else if (sfe.FixAddCurve3d (edge)) {
    //Creates and adds a 3D curve to the edge
  }
  sfe.FixSameParameter (edge);}
```

2.3.9. Repairing tool for the wireframe of a shape (Class ShapeFix_Wireframe)

This tool provides methods for geometrical fixing of gaps and merging small edges in a shape. This class performs the following operations:

- fills gaps in the 2D and 3D wireframe of a shape.
- merges and removes small edges.

Fixing of small edges can be managed with the help of two flags:

- *ModeDropSmallEdges()* – mode for removing small edges that can not be merged, by default it is equal to Standard_False.
- *LimitAngle* – maximum possible angle for merging two adjacent edges, by default no limit angle is applied (-1).

To perform fixes it is necessary to:

- create a tool and initialize it by shape,
- set the working precision problems will be detected with and the maximum allowed tolerance
- perform fixes

Example:

```
//creation of a tool
Handle(ShapeFix_Wireframe) sfwf = new ShapeFix_Wireframe(shape);
//sets the working precision problems will be detected with and
//the maximum allowed tolerance
sfwf->SetPrecision(prec);
sfwf->SetMaxTolerance(maxTol);
//fixing of gaps
sfwf->FixWireGaps();
//fixing of small edges
```

```
//setting of the drop mode for the fixing of small edges and max
possible angle between merged edges.
sfwf->ModeDropSmallEdges = Standard_True;
sfwf->SetLimliteAngle(angle);
//performing the fix
sfwf->FixSmallEdges();
//getting the result
TopoDS_Shape resShape = sfwf->Shape();
```

It is desirable that a shape is topologically correct before applying the methods of this class.

2.3.10. Tool for removing small faces from a shape (Class ShapeFix_FixSmallFace)

This tool is intended for dropping small faces from the shape. The following cases are processed:

spot face : if the size of the face is less than the given precision,

strip face: if the size of the face in one dimension is less then the given precision.

The sequence of actions for performing the fix is the same as for fixes described above:

Example:

```
//creation of a tool
Handle(ShapeFix_FixSmallFace) sff = new
ShapeFix_FixSmallFace(shape);
//setting of tolerances
sff->SetPrecision(prec);
sff->SetMaxTolerance(maxTol);
//performing fixes
sff.Perform();
//getting the result
TopoDS_Shape resShape = sff.FixShape();
```

2.3.11. Tool to modify tolerances of shapes (Class ShapeFix_ShapeTolerance).

This tool provides a functionality to set tolerances of a shape and its sub-shapes.

In Open CASCADE only the following shapes have tolerances:

vertices,

edges,

faces.

This tool allows processing each concrete type of sub-shapes or all types at a time.

You set the tolerance functionality as follows:

- set a tolerance for sub-shapes, by method SetTolerance,
- limit tolerances with given ranges, by method LimitTolerance.

Example:

```
//creation of a tool
ShapeFix_ShapeTolerance Sft;
//setting a specified tolerance on shape and all of its sub-shapes.
Sft.SetTolerance(shape,toler);
//setting a specified tolerance for vertices only
Sft.SetTolerance(shape,toler,TopAbs_VERTEX);
//limiting the tolerance on the shape and its sub-shapes between
minimum and
//maximum tolerances
Sft.LimitTolerance(shape,tolermin,tolermax);
```

3. *Analysis*

3.1. Overview.

The ShapeAnalysis package contains various tools for the analysis of topological shapes.

At present, there are:

methods and tools for the analysis of validity of shapes which are used in the repairing tools:

- method to check the orientation of face boundaries (method ShapeAnalysis::IsOuterBound)
- tool to check the validity of wires (class ShapeAnalysis_Wire)
- tool to check the validity of edges (class ShapeAnalysis_Edge)
- tool to check the presence of small faces in the shape (class ShapeAnalysis_CheckSmallFace)
- tool for an analysis of orientation of edges in a shell and an analysis of a closure of shell.
- tools to analyze shape properties
- tool to analyze shape tolerances (ShapeAnalysis_ShapeTolerance)
- tool to analyze free boundaries of a shape (ShapeAnalysis_FreeBounds)
- tool to analyze the contents of a shape (ShapeAnalysis_ShapeContents)

3.2. Short description of tools for the analysis of validity of shapes.

It is not necessary to check a shape by these tools before the execution of repairing tools because these tools are used for the analysis before performing fixes inside the repairing tools.

However, if you want, these tools can be used for detecting some of shape problems independently from the repairing tools.

It can be done in the following way:

- create an analysis tool.
- initialize it by shape and set a tolerance problems will be detected with if it is necessary.
- check the problem that interests you.

Example:

```
TopoDS_Face face = ...;
ShapeAnalysis_Edge sae;
//Creates a tool for analyzing an edge
for(TopExp_Explorer Exp(face,TopAbs_EDGE);Exp.More();Exp.Next()) {
  TopoDS_Edge edge = TopoDS::Edge (Exp.Current());
  if (!sae.HasCurve3d (edge)) {
    cout << “Edge has no 3D curve” << endl;  }
}
```

3.2.1. Method for the analysis of orientation wires on a face.

It is possible to check whether a face has an outer boundary with the help of method ShapeAnalysis::IsOuterBound.

Example:

```
TopoDS_Face face … //analyzed face
if(!ShapeAnalysis::IsOuterBound(face)) {
       cout<<”Face has not outer boundary””<<endl;
}
```

3.2.2. Tools for the analysis of validity of wires (Class ShapeAnalysis_Wire)

This class is intended to analyze a wire. It provides functionalities both to explore wire properties and to check its conformance to Open CASCADE requirements.

These functionalities include:

- checking the order of edges in the wire,
- checking for the presence of small edges (with a length less than the given value),
- checking for the presence of disconnected edges (adjacent edges having different vertices),
- checking the consistency of edge curves,
- checking for the presence or missing of degenerated edges,
- checking for the presence of self-intersecting edges and intersecting edges (edges intersection is understood as intersection of their 2D curves),
- checking for lacking edges to fill gaps in the surface parametrical space,
- analyzing the wire orientation (to define the outer or the inner bound on the face),
- analyzing the orientation of the shape (edge or wire) being added to an already existing wire.

Please NOTE

All checking operations except for the first one are based on the assumption that edges in the wire are ordered. Thus, if the wire is detected as non-ordered it is necessary to order it before calling other checking operations. This can be done, for example, with the help of the ShapeFix_Wire::FixOrder() method.

This tool should be initialized with the following data:

wire,

face (or a surface with a location),

precision.

Once the tool has been initialized, it is possible to perform the necessary checking operations. In order to obtain all information on a wire at a time the global method *Perform* is provided. It calls all other API checking operations to check each separate case.

API methods check for corresponding cases only, the value and the status they return can be analyzed to understand whether the case was detected or not.

Some methods in this class are:

• *CheckOrder* checks whether edges in the wire are in the right order

• *CheckConnected* checks whether edges are disconnected

• *CheckSmall* checks whether there are edges that are shorter than the given value

• *CheckSelfIntersection* checks, whether there are self-intersecting or adjacent intersecting edges. If the intersection takes place due to nonadjacent edges, it is not detected.

This class maintains status management. Each API method stores the status of its last execution which can be queried by the corresponding Status..() method. In addition, each API method returns a Boolean value, which is True when a case being analyzed is detected (with the set ShapeExtend_DONE status), otherwise it is False.

Example:

```
TopoDS_Face face = ...;
TopoDS_Wire wire = ...;
Standard_Real precision = 1e-04;
ShapeAnalysis_Wire saw (wire, face, precision);
//Creates a tool and loads objects into it
if (saw.CheckOrder()) {
  cout<<“Some edges in the wire need to be reordered”<<endl;
  cout<<“Please ensure that all the edges are correctly
             ordered before further analysis”<<endl;
  return;
}
if (saw.CheckSmall (precision)) {
  cout<<“Wire contains edge(s) shorter than “<<precision<<endl;
}
if (saw.CheckConnected()) {
  cout<<“Wire is disconnected”<<endl;
}
if (saw.CheckSelfIntersection()) {
  cout<<“Wire has self-intersecting or intersecting
             adjacent edges” << endl;
}
```

3.2.3. Tool for checking the validity of edges (Class ShapeAnalysis_Edge)

This class is intended to analyze edges. It provides the following functionalities to work with an edge:

- querying geometrical representations (3D curve and pcurve(s) on a given face or surface),
- querying topological sub-shapes (bounding vertices),
- checking overlapping edges,
- analyzing the curves consistency:
  - mutual orientation of the 3D curve and 2D curve (co-directions or opposite directions),
  - correspondence of 3D and 2D curves to vertices.

This class supports status management described above.

Example:

```
TopoDS_Face face = ...;
```

```
ShapeAnalysis_Edge sae;
//Creates a tool for analyzing an edge
for(TopExp_Explorer Exp(face,TopAbs_EDGE);Exp.More();Exp.Next()) {
  TopoDS_Edge edge = TopoDS::Edge (Exp.Current());
  if (!sae.HasCurve3d (edge)) {
    cout << “Edge has no 3D curve” << endl;
  }
  Handle(Geom2d_Curve) pcurve;
  Standard_Real cf, cl;
  if (sae.PCurve (edge, face, pcurve, cf, cl, Standard_False)) {
    //Returns the pcurve and its range on the given face
    cout<<“Pcurve range [“<<cf<<“, “<<cl<<“]”<< endl;
  }
  Standard_Real maxdev;
  if (sae.CheckSameParameter (edge, maxdev)) {
    //Checks the consistency of all the curves
    //in the edge
    cout<<“Incorrect SameParameter flag”<<endl;
  }
  cout<<“Maximum deviation “<<maxdev<<“, tolerance“
             <<BRep_Tool::Tolerance(edge)<<endl;
}
//checks the overlapping of two edges
if(sae.CheckOverlapping(edge1,edge2,prec,dist)) {
 cout<<”Edges are overlapped with tolerance = ”<<prec<<endl;
 cout<<”Domain of overlapping =”<<dist<<endl;
}
```

3.2.4. Tool for the analysis of presence of small faces (Class ShapeAnalysis_CheckSmallFace)

This class is intended for analyzing small faces from the shape. The following cases are processed:

spot face : if the size of the face is less than the given precision, (method CheckSpotFace())

strip face : if the size of the face in one dimension is less than the given precision. (method CheckStripFace)

Example:

```
TopoDS_Shape shape … // checked shape
//Creation of a tool
ShapeAnalysis_CheckSmallFace saf;
//exploring the shape on faces and checking each face
Standard_Integer numSmallfaces =0;
```

```
for(TopExp_Explorer aExp(shape,TopAbs_FACE); aExp.More();
aExp.Next()) {
 TopoDS_Face face = TopoDS::Face(aexp.Current());
 TopoDS_Edge E1,E2;
 if(saf.CheckSpotFace(face,prec) ||
       saf.CheckStripFace(face,E1,E2,prec))
      NumSmallfaces++;
}
if(numSmallfaces)
 cout<<"Number of small faces in the shape ="<< numSmallfaces
<<endl;
```

3.2.5. Tools for the analysis of validity of a shell and its closure (Class ShapeAnalysis_Shell).

This tool is intended to check the orientation of edges in a manifold shell. With the help of this tool, free edges (edges entered into one face) and bad edges (edges entered into the shell twice with the same orientation) can be found. By occurrence of bad and free edges a conclusion about the shell validity and the closure of the shell can be made.

Example:

```
TopoDS_Shell shell // checked shape
ShapeAnalysis_Shell sas(shell);
//analysis of the shell , second parameter is set to True for
//getting free edges,(default False)
sas.CheckOrientedShells(shell,Standard_True);
//getting the result of analysis
if(sas.HasBadEdges()) {
cout<<" Shell is invalid<<endl;
TopoDS_Compound badEdges = sas.BadEdges();
}
if(sas.HasFreeEdges()) {
 cout<<"Shell is open"<<endl;
 TopoDS_Compound freeEdges = sas.FreeEdges();
}
```

3.3. Short description of tools for analysis of properties of shapes.

3.3.1. Tool for analysis of tolerance on shape (Class ShapeAnalysis_ShapeTolerance)

This class is intended to compute tolerances of the shape and its sub-shapes.

In Open CASCADE only the following shapes have tolerances:

vertices,

edges,

faces.

This tool allows to analyze each concrete type of sub-shapes or all types at a time.

The analysis of tolerance functionality is the following:

- computing the minimum, maximum and average tolerances of sub-shapes,
- finding sub-shapes with tolerances exceeding the given value,
- finding sub-shapes with tolerances in the given range.

Example:

```
TopoDS_Shape shape = ...;
ShapeAnalysis_ShapeTolerance sast;
Standard_Real AverageOnShape = sast.Tolerance (shape, 0);
cout<<“Average tolerance of the shape is “<<AverageOnShape<<endl;
Standard_Real MinOnEdge = sast.Tolerance (shape,-1,TopAbs_EDGE);
cout<<“Minimum tolerance of the edges is “<<MinOnEdge<<endl;
Standard_Real MaxOnVertex = sast.Tolerance (shape,1,TopAbs_VERTEX);
cout<<“Maximum tolerance of the vertices is “<<MaxOnVertex<<endl;
Standard_Real MaxAllowed = 0.1;
if (MaxOnVertex > MaxAllowed) {
  cout<<“Maximum tolerance of the vertices exceeds
                   maximum allowed”<<endl;
}
```

3.3.2. Tool for the analysis of free boundaries. (Class ShapeAnalysis_FreeBounds)

This class is intended to analyze and output the free bounds of the shape. Free bounds are wires consisting of edges referenced only once by the only face in the shape.

This class works on two distinct types of shapes when analyzing their free bounds:

- Analysis of possible free bounds taking the specified tolerance into account. This analysis can be applied to a compound of faces. The analyzer of the sewing algorithm (BRepAlgo_Sewing) is used to forecast what free bounds would be obtained after the sewing of these faces is performed. The following method should be used for this analysis:

  ```
  ShapeAnalysis_FreeBounds safb(shape,toler);
  ```

- Analysis of already existing free bounds. Actual free bounds (edges shared by the only face in the shell) are output in this case. ShapeAnalysis_Shell is used for that.

  ```
  ShapeAnalysis_FreeBounds safb(shape);
  ```

When connecting edges into wires this algorithm tries to build wires of maximum length. Two options are provided for the user to extract closed sub-contours out of closed and/or open contours. Free bounds are returned as two compounds, one for closed and one for open wires. To obtain a result it is necessary to use methods:

```
TopoDS_Compound ClosedWires  = safb.GetClosedWires();
TopoDS_Compound OpenWires = safb.GetOpenWires();
```

This class also provides some static methods for advanced use: connecting edges/wires to wires, extracting closed sub-wires from wires, distributing wires into compounds for closed and open wires.

Example:

```
TopoDS_Shape shape = ...;
Standard_Real SewTolerance = 1.e-03;
//Tolerance for sewing
Standard_Boolean SplitClosed = Standard_False;
Standard_Boolean SplitOpen = Standard_True;
//in case of analysis of possible free boundaries
ShapeAnalysis_FreeBounds safb (shape, SewTolerance,
             SplitClosed, SplitOpen);
//in case of analysis of existing free bounds
ShapeAnalysis_FreeBounds safb (shape, SplitClosed, SplitOpen);
//getting the results
TopoDS_Compound ClosedWires = safb.GetClosedWires();
//Returns a compound of closed free bounds
TopoDS_Compound OpenWires = safb.GetClosedWires();
//Returns a compound of open free bounds
```

3.3.3. Tool for the analysis of shape contents (Class ShapeAnalysis_ShapeContents).

This class provides tools counting the number of sub-shapes and selecting a sub-shape by the following criteria:

Methods for getting the number of sub-shapes:

- number of solids,
- number of shells,
- number of faces,
- number of edges,
- number of vertices.

Methods for calculating the number of geometrical objects or sub-shapes with a specified type:

- number of free faces,
- number of free wires,
- number of free edges,
- number of C0 surfaces,
- number of C0 curves,
- number of BSpline surfaces,… etc

and selecting sub-shapes by various criteria. Corresponding flags should be set to True for storing a shape by a specified criteria:

- faces based on indirect surfaces,

```
safc.MofifyIndirectMode() = Standard_True;
```

- faces based on offset surfaces,

```
safc.ModifyOffsetSurfaceMode() = Standard_True;
```

- edges if their 3D curves are trimmed,

```
safc.ModifyTrimmed3dMode() = Standard_True;
```

- edges if their 3D curves and 2D curves are offset curves,

```
safc.ModifyOffsetCurveMode() = Standard_True;
```

- edges if their 2D curves are trimmed.

```
safc.ModifyTrimmed2dMode() = Standard_True;
```

Example:

```
Selecting faces based on offset surfaces.
ShapeAnalysis_ShapeContents safc;
//set a corresponding flag for storing faces based on the offset
surfaces
safc.ModifyOffsetSurfaceMode() = Standard_True;
safc.Perform(shape);
//getting the number of offset surfaces in the shape
Standard_Integer NbOffsetSurfaces = safc.NbOffsetSurf();
//getting the sequence of faces based on offset surfaces.
Handle(TopTools_HSequenceOfShape) seqFaces =
safc.OffsetSurfaceSec();
```

4. *Upgrading*

Upgrading tools are intended for adaptation of shapes for better use by Open CASCADE or for customization to particular user's needs, i.e. for export to another system. This means that not only it corrects and upgrades but also changes the definition of a shape with regard to its geometry, size and other aspects. Convenient API allows you to create your own tools to perform specific upgrading. Additional tools for particular cases provide an ability to divide shapes and surfaces according to certain criteria.

4.1. Tools for splitting a shape according to a specified criterion.

4.1.1. Overview.

These tools provide such modifications when one topological object can be divided or converted to several ones according to specified criteria. Besides, there are high level API tools for particular cases which:

- Convert the geometry of shapes up to a given continuity,
- split revolutions by U to segments less than the given value,
- convert to Bezier surfaces and Bezier curves,
- split closed faces,
- convert C0 BSpline curve to a sequence of C1 BSpline curves.

All tools for particular cases are based on general tools for shape splitting but each of them has its own tools for splitting or converting geometry in accordance with the specified criteria.

General tools for shape splitting are:

- tool for splitting the whole shape,
- tool for splitting a face,
- tool for splitting wires.

Tools for shape splitting use tools for geometry splitting:

- tool for splitting surfaces,
- tool for splitting 3D curves,
- tool for splitting 2D curves.

4.1.2. Using tools available for shape splitting.

If it is necessary to split a shape by a specified continuity, split closed faces in the shape, split surfaces of revolution in the shape by angle or to convert all surfaces, all 3D curves, all 2D curves in the shape to Bezier, it is possible to use the existing/available tools.

The usual way to use these tools exception for the tool of converting a C0 BSpline curve is the following:

- a tool is created and initialized by shape.
- work precision for splitting and the maximum allowed tolerance are set
- the value of splitting criterion Is set (if necessary)
- splitting is performed.
- splitting statuses are obtained.

- result is obtained
- the history of modification of the initial shape and its sub-shapes is output (this step is optional).

Example.

Splitting all surfaces and all 3D and 2D curves having a continuity of less the C2.

```
//creates a tool and initializes it by shape.
ShapeUpgrade_ShapeDivideContinuity ShapeDivedeCont(initShape);

//sets the working 3D and 2D precision and the maximum allowed
//tolerance
ShapeDivideCont.SetTolerance(prec);
ShapeDivideCont.SetTolerance2D(prec2d);
ShapeDivideCont.SetMaxTolerance(maxTol);

//sets the values of criteria for surfaces, 3D curves and 2D curves.
ShapeDivideCont.SetBoundaryCriterion(GeomAbs_C2);
ShapeDivideCont.SetPCurveCriterion(GeomAbs_C2);
ShapeDivideCont.SetSurfaceCriterion(GeomAbs_C2);

//performs the splitting.
ShapeDivideCont.Perform();

//checks the status and gets the result
if(ShapeDivideCont.Status(ShapeExtend_DONE)
 TopoDS_Shape result = ShapeDivideCont.GetResult();
//gets the history of modifications made to faces
for(TopExp_Explorer aExp(initShape,TopAbs_FACE); aExp.More(0;
aExp.Next()) {
  TopoDS_Shape modifShape = ShapeDivideCont.GetContext()->
                             Apply(aExp.Current());
}
```

4.1.3. Creation of a new tool for splitting a shape.

To create a new splitting tool it is necessary to create tools for geometry splitting according to a desirable criterion. These new tools should be inherited from basic tools for geometry splitting. Then these new tools should be set into corresponding tools for shape splitting.

- a new tool for surface splitting should be set into the tool for face splitting
- new tools for splitting of 3D and 2D curves should be set into the splitting tool for wires.

In order to be able to change the value of criterion of shape splitting it is necessary to create a new tool for shape splitting that should be inherited from the general splitting tool for shapes.

Example:

Splitting a shape according to a specified criterion.

```
//creation of new tools for geometry splitting by a specified
//criterion.
Handle(MyTools_SplitSurfaceTool) MySplitSurfaceTool =
                                     new MyTools_SplitSurfaceTool;
Handle(MyTools_SplitCurve3DTool) MySplitCurve3Dtool =
                                     new MyTools_SplitCurve3DTool;
Handle(MyTools_SplitCurve2DTool) MySplitCurve2Dtool =
                                     new MyTools_SplitCurve2DTool;

//creation of  a tool for splitting the shape and initialization of
//that tool by shape.
TopoDS_Shape initShape
MyTools_ShapeDivideTool ShapeDivide (initShape);

//setting of work precision for splitting and maximum allowed
//tolerance.
ShapeDivide.SetPrecision(prec);
ShapeDivide.SetMaxTolerance(MaxTol);

//setting of new splitting geometry tools in the shape splitting
//tools
Handle(ShapeUpgrade_FaceDivide) FaceDivide =
                               ShapeDivide->GetSplitFaceTool();
Handle(ShapeUpgrade_WireDivide) WireDivide =
                               FaceDivide->GetWireDivideTool();
FaceDivide->SetSplitSurfaceTool(MySplitSurfaceTool);
WireDivide->SetSplitCurve3dTool(MySplitCurve3DTool);
WireDivide->SetSplitCurve2dTool(MySplitCurve2DTool);

//setting of the value criterion.
 ShapeDivide.SetValCriterion(val);

//shape splitting
ShapeDivide.Perform();

//getting the result
TopoDS_Shape splitShape = ShapeDivide.GetResult();

//getting the history of modifications of faces
for(TopExp_Explorer aExp(initShape,TopAbs_FACE); aExp.More(0;
aExp.Next()) {
TopoDS_Shape modifShape = ShapeDivide.GetContext()->
```

```
                                        Apply(aExp.Current());
    }
```

4.1.4. Short description of general splitting tools.

General tool for shape splitting (Class ShapeUpgrade_ShapeDivide)

This tool provides shape splitting and converting according to the given criteria. It performs these operations for each face with the given tool for face splitting (ShapeUpgrade_FaceDivide by default).

This tool provides access to the tool for dividing faces with the help of the following methods:

SetSplitFaceTool,

GetSpliFaceTool.

General tool for face splitting (Class ShapeUpgrade_FaceDivide)

This tool divides a Face (edges in the wires, by splitting 3D and 2D curves, as well as the face itself, by splitting the supporting surface) according to the given criteria.

The area of the face intended for division is defined by 2D curves of the wires on the Face.

All 2D curves are supposed to be defined (in the parametric space of the supporting surface).

The result is available after the call to the *Perform* method. It is a Shell containing all resulting Faces. All modifications made during the splitting operation are recorded in the external context (ShapeBuild_ReShape).

This tool provides access to the tool for wire division and surface splitting by means of methods:

SetWireDivideTool,

GetWireDivideTool,

SetSurfaceSplitTool,

GetSurfaceSplitTool.

General tool for wire splitting (Class ShapeUpgrade_WireDivide)

This tool divides edges in the wire lying on the face or free wires or free edges with a given criterion. It splits the 3D curve and 2D curve(s) of the edge on the face. Other 2D curves, which may be associated with the edge, are simply copied. If the 3D curve is split then the 2D curve on the face is split as well, and vise-versa. The original shape is not modified. Modifications made are recorded in the context (ShapeBuild_ReShape).

This tool provides access to the tool for dividing and splitting 3D and 2D curves by means of methods:

SetEdgeDivdeToo,l

GetEdgeDivideTool,

SetSplitCurve3dTool,

GetSplitCurve3dTool,

SetSplitCurve2dTool,

GetSplitCurve2dTool

and it also provides access to the mode for splitting edges by methods:

SetEdgeMode,

GetEdgeMode

This mode manages whether only free edges, only shared edges or all edges are split.

General tool for edge splitting. (ShapeUpgrade_EdgeDivide)

This tool divides edges and their geometry according to the specified criteria. It is used in the wire-dividing tool.

This tool provides access to the tool for dividing and splitting 3D and 2D curves by the following methods:

SetSplitCurve3dTool,

GetSplitCurve3dTool,

SetSplitCurve2dTool,

GetSplitCurve2dTool

General tools for geometry splitting.

There are three general tools for geometry splitting.

- General tool for surface splitting.(***ShapeUpgrade_SplitSurface***)
- General tool for splitting 3D curves.(***ShapeUpgrade_SplitCurve3d***)
- General tool for splitting 2D curves.(***ShapeUpgrade_SplitCurve2d***)

All these tools are constructed the same way:

They have methods:

- for initializing by geometry (method *Init*)
- for splitting (method *Perform*)
- for getting the status after splitting and the results:

  Status – for getting the result status,

  ResSurface - for splitting surfaces.

  GetCurves - for splitting 3D and 2D curves.

During the process of splitting in the method *Perform* :

- splitting values in the parametric space are computed according to a specified criterion (method *Compute*)
- splitting is made in accordance with the values computed for splitting (method *Build*).

To create new tools for geometry splitting it is enough to inherit a new tool from the general tool for splitting a corresponding type of geometry and to re-define the method for computation of splitting values according to the specified criterion in them. (method *Compute*).

Example.

Header file for the tool for surface splitting by continuity:

```
class ShapeUpgrade_SplitSurfaceContinuity : public
ShapeUpgrade_SplitSurface {

Standard_EXPORT ShapeUpgrade_SplitSurfaceContinuity();


//methods to set the criterion and the tolerance into the splitting
//tool

Standard_EXPORT   void SetCriterion(const GeomAbs_Shape Criterion) ;
Standard_EXPORT   void SetTolerance(const Standard_Real Tol) ;

//re-definition of method Compute
Standard_EXPORT virtual void Compute(const Standard_Boolean Segment)
;
Standard_EXPORT ~ShapeUpgrade_SplitSurfaceContinuity();
private:
GeomAbs_Shape myCriterion;
Standard_Real myTolerance;
Standard_Integer myCont;
};
```

4.1.5. Short description of existing splitting tools.

Tool for shape geometry conversion to target continuity

(Class ShapeUpgrade_ShapeDivideContinuity)

This tool allows converting geometry with continuity less than the specified continuity to geometry with target continuity. If converting is not possible than geometrical object is split into several ones, which satisfy the given criteria. A topological object based on this geometry is replaced by several objects based on the new geometry.

Example

```
ShapeUpgrade_ShapeDivideContinuity sdc (shape);
sdc.SetTolerance (tol3d);
sdc.SetTolerance3d (tol2d); // if known, else 1.e-09 is taken
sdc.SetBoundaryCriterion (GeomAbs_C2); // for Curves 3D
sdc.SetPCurveCriterion (GeomAbs_C2); // for Curves 2D
sdc.SetSurfaceCriterion (GeomAbs_C2); // for Surfaces
sdc.Perform ();
TopoDS_Shape bshape = sdc.Result();
.. to also get the correspondances before/after
Handle(ShapeBuild_ReShape) ctx = sdc.Context();
.. on a given shape
if (ctx.IsRecorded (sh)) {
  TopoDS_Shape newsh = ctx->Value (sh);
// if there are several results, they re-recorded inside a
//Compound
.. process as needed
}
```

Tool for splitting by angle (Class ShapeUpgrade_ShapeDivideAngle)

This tool is intended for splitting all surfaces of revolution, cylindrical, toroidal, conical, spherical surfaces in the given shape so that each resulting segment covers not more than the defined angle (in radians).

Tool for conversion 2D, 3D curves and surfaces to Bezier

(Class ShapeUpgrade_ShapeConvertToBezier)

API tool for performing a conversion of 3D, 2D curves to Bezier curves and surfaces to Bezier based surfaces (Bezier surface, surface of revolution based on Bezier curve, offset surface based on any of previous types).

This tool provides access to various flags for conversion of different types of curves and surfaces to Bezier by methods:

For 3D curves:

Set3dConversion,

Get3dConversion,

Set3dLineConversion,

Get3dLineConversion,

Set3dCircleConversion,

Get3dCircleConversion,

Set3dConicConversion,

Get3dConicConversion

For 2D curves:

Set2dConversion,

Get2dConversion

For surfaces :

GetSurfaceConversion,

SetPlaneMode,

GetPlaneMode,

SetRevolutionMode,

GetRevolutionMode,

SetExtrusionMode,

GetExtrusionMode,

SetBSplineMode,

GetBSplineMode,

Example

Conversion of planes to Bezier surfaces.

```
//Creation and initialization of a tool.
ShapeUpgrade_ShapeConvertToBezier SCB (Shape);
//setting tolerances
…..
//setting mode for conversion of planes
SCB.SetSurfaceConversion (Standard_True);
SCB.SetPlaneMode(Standard_True);
SCB.Perform();
If(SCB.Status(ShapeExtend_DONE)
    TopoDS_Shape result = SCB.GetResult();
```

Tool for splitting closed faces (Class ShapeUpgrade_ShapeDivideClosed)

This tool provides the splitting of closed faces in the shape to a defined number of components by the U and V parameters. It topologically and (partially) geometrically processes closed faces and performs splitting with the help of class ShapeUpgrade_ClosedFaceDivide.

Example

```
TopoDS_Shape aShape = …;
ShapeUpgrade_ShapeDivideClosed tool (aShape );
Standard_Real closeTol = …;
tool.SetPrecision(closeTol);
Standard_Real maxTol = …;
tool.SetMaxTolerance(maxTol);
Standard_Integer NbSplitPoints = …;
tool.SetNbSplitPoints(num);
if ( ! tool.Perform() && tool.Status (ShapeExtend_FAIL) ) {
  cout<<"Splitting of closed faces failed"<<endl;
  . . .
}
TopoDS_Shape aResult = tool.Result();
```

Tool for splitting a C0 BSpline 2D or 3D curve to a sequence C1 BSpline curves

The API methods for this tool is a package of methods :

ShapeUpgrade::C0BSplineToSequenceOfC1BsplineCurve

Converts a C0 B-Spline curve into a sequence of C1 B-Spline curves. This method splits a B-Spline at the knots with multiplicities equal to degree, it does not use any tolerance and therefore does not change the geometry of the B-Spline. Returns True if C0 B-Spline was successfully split, otherwise returns False (if BS is C1 B-Spline).

Tool for splitting faces.

Overview

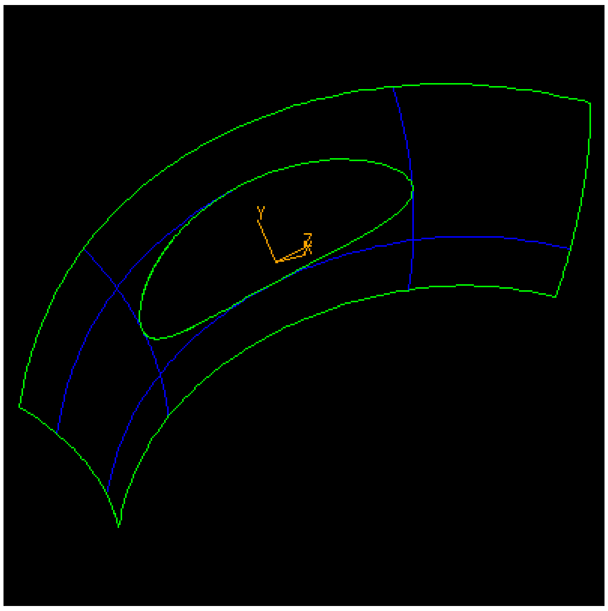
This tool is implemented as a new ShapeHealing operator in the package ShapeUpgrade. ShapeUpgrade_ShapeDivideArea. This tool can work with various types of shapes:

- Compound,
- Solid,
- Shell,
- Face.

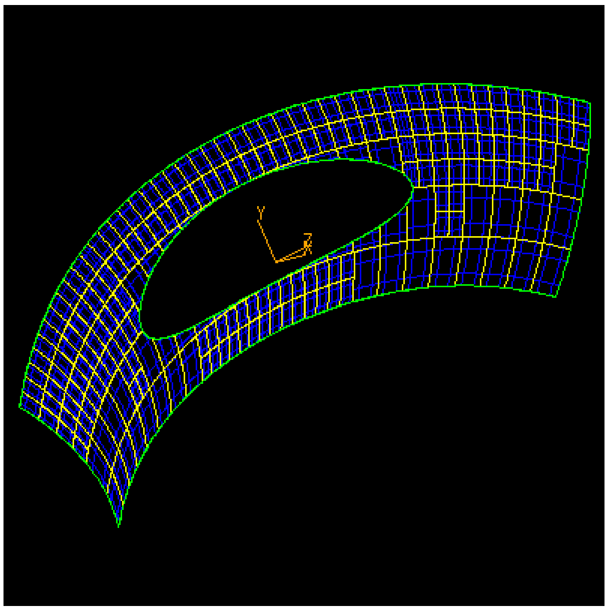
In the process of work this tool examines each face of a specified shape and if the face area exceeds the specified maximal area, this face is divided. Face splitting is performed in the parametric space of this face. The values of splitting in U and V directions are calculated with the account of translation of the bounding box form parametric space to 3D space. Such calculations are necessary to avoid creation of strip faces. In the process of splitting the holes on the initial face are taken into account. After the splitting all new faces are checked by area again and the splitting procedure is repeated for the faces whose area still exceeds the max allowed area. Sharing between faces in the shape is preserved and the resulting shape is of the same type as the source shape.

An example of using this tool is presented in the figures below:

Source face



Result of splitting tool (obtained shape is SHELL)



Description of the implemented classes.

All necessary classes are implemented in the package **ShapeUpgrade**.

API class ShapeUpgade_ShapeDivideArea

This class is inherited from the base class **ShapeUpgrade_ShapeDivide**.

- This class should be initialized on a shape with the help of the constructor or the method Init() from the base class.
- The maximal allowed area should be specified by the method MaxArea().
- To produce a splitting use the method Perform from the base class
- The result shape can be obtained with the help the method Result().

Example:

```
ShapeUpgrade_ShapeDivideArea tool (inputShape);
tool.MaxArea() = aMaxArea;
tool.Perform();
if(tool.Status(ShapeExtend_DONE)) {
  TopoDS_Shape ResultShape = tool.Result();
  ShapeFix::SameParameter ( ResultShape, Standard_False );
}
```

Please note that the use of method ShapeFix::SameParameter is necessary, otherwise the parameter edges obtained as a result of splitting can be different .

Private classes

- Class **ShapeUpgrade_FaceDivideArea** is inherited from ShapeUpgrade_FaceDivide.

  This class is intended for splitting a face by the maximal area criterion.
- Class **ShapeUpgrade_SplitSurfaceArea** is inherited from ShapeUpgrade_SplitSurface.

  This class calculates the parameters of face splitting in the parametric space.

Draw command.

DRAW command "**DT_SplitByArea**" is implemented for the tool in **class SWDRAW_ShapeUpgrade**.

Command format:

DT_SplitByArea result_shape initial_shape max_area [preci]

Command parameters:

Mandatory parameters:

- result_shape – the resulting shape obtained as a result of splitting.
- intial_shape – the shape splitting can be performed on.
- max_area – the maximal allowed area for a face.

Optional parameters:

- preci – precision used for calculation of faces area.(by default this value is equal to 1.e-5).

4.2. Customization of shapes.

4.2.1. Overview

Customization tools are intended for adaptation of shape geometry in compliance with the customer needs. These tools provide modifications of one geometrical object to another geometrical object in the shape. The following modifications are supported:

1. Modification of shape geometry:
    - conversion of indirect elementary surfaces (with left-handed coordinate systems) in the shape into direct ones,
    - conversion of elementary surfaces in the shape to surfaces of revolution,
    - conversion of surfaces of linear extrusion, revolution, offset surfaces to BSpline surfaces,
    - modification of parameterization, degree, number of segments of BSpline surfaces and BSpline curves and rational BSpline surfaces. Conversion of all types of surfaces and curves to BSpline surfaces with a required degree or number of spans and continuity.
    - scaling the shape
2. Modification of a surface
    - conversion of BSpline and Bezier surfaces to analytical form,

4.2.2. Short description of tools for modification of shapes.

All tools for shape geometry modification are implemented in a similar way through a set of methods of package ShapeCustom. These tools are provided for:

- converting indirect surfaces ***(ShapeCustom::DirectFaces())***
- modification BSpline curves and surfaces and converting other surfaces and curves to BSpline with required parameters ***(ShapeCustom::BSplineRestriction())***
- conversion of elementary surfaces in the shape to surfaces of revolution ***(ShapeCustom::ConvertToRevolution())***
- conversion of surface of linear extrusion, revolution, offset surface to BSpline surfaces ***(ShapeCustom::ConvertToBSpline())***
- scaling the shape (***ShapeCustom::ScaleShape(***))

To ensure the necessary modification of a shape it is enough to initialize the appropriate tool by the shape and desirable parameters and to get the resulting shape.

Example:

Conversion of indirect surfaces in the shape.

```
TopoDS_Shape initialShape ..
TopoDS_Shape resultShape = ShapeCustom::DirectFaces(initialShape);
```

<u>Conversion of indirect surfaces in the shape.</u>

ShapeCustom::DirectFaces

```
static TopoDS_Shape DirectFaces(const TopoDS_Shape& S);
```

This class returns a new shape without indirect surfaces (elementary surfaces with a left-handed coordinate system).

It uses the tool *ShapeCustom_DirectModification*.

Please NOTE:

New 2d curves (recomputed for converted surfaces) are added to the same edges being shared by both the resulting shape and the original shape <S>.

Scale the shape.

ShapeCustom::ScaleShape

```
TopoDS_Shape ShapeCustom::ScaleShape(const TopoDS_Shape& S,
        const Standard_Real scale);
```

Returns a new shape, which is a scaled original shape with a coefficient equal to the specified value of scale.

It uses the tool *ShapeCustom_TrsfModification.*

Modification of BSline curves and surface and conversion of other type curves and surfaces to BSpline.

This tool is intended for the approximation of surfaces, curves and 2D curves with a specified degree, max number of segments, 2d tolerance, 3d tolerance. If the approximation result cannot be achieved with the specified continuity, the latter can be reduced.

It uses the tool *ShapeCustom_BSplineRestriction.*

ShapeCustom::BsplineRestriction

```
TopoDS_Shape ShapeCustom::BSplineRestriction (const TopoDS_Shape& S,
        const Standard_Real Tol3d, const Standard_Real Tol2d,
        const Standard_Integer MaxDegree,
        const Standard_Integer MaxNbSegment,
        const GeomAbs_Shape Continuity3d,
        const GeomAbs_Shape Continuity2d,
        const Standard_Boolean Degree,
        const Standard_Boolean Rational,
        const Handle(ShapeCustom_RestrictionParameters)& aParameters)
```

Returns a new shape with all surfaces, curves and 2D curves which type is BSpline/Bezier or based on the two latter, converted with Degree less than <MaxDegree> or with a number of spans less than <NbMaxSegment> depending on priority parameter <Degree>. If this parameter is equal to True then Degree will be increased to the value <GmaxDegree>, otherwise the NbMaxSegments will be increased to the value <GmaxSegments>. <GmaxDegree> and <GMaxSegments> are the maximum possible degree and the number of spans correspondingly. These values will be used in those cases when an approximation with specified parameters is impossible and one of GmaxDegree or GMaxSegments is selected depending on priority.

Note that if approximation is impossible with <GMaxDegree>, even then the number of spans can exceed the specified <GMaxSegment>. <Rational> specifies whether Rational BSpline/Bezier should be converted into polynomial B-Spline.

Please NOTE:

The continuity of surfaces in the resulting shape can be less than the given value.

In order to convert other types of curves and surfaces to BSpline with required parameters it is necessary to use flags from class ShapeCustom_RestrictionParameters, which is just a container of flags.

Flags to define whether a specified-type geometry has been converted to BSpline with the required parameters:

ConvertPlane,

ConvertBezierSurf,

ConvertRevolutionSurf

ConvertExtrusionSurf,.

ConvertOffsetSurf,

ConvertCurve3d,

ConvertOffsetCurv3d,

ConvertCurve2d,

ConvertOffsetCurv2d,

SegmentSurfaceMode

Please NOTE:

Parameters ConvertCurve3d and ConvertCurve2d are responsible for the conversion of all types of 3D and 2D curves. But parameters ConvertOffsetCurv3d, ConvertOffsetCurv2d are responsible for conversion of offset 3D and 2D curves. Parameter SegmentSurfaceMode defines whether the surface would be approximated within the boundaries of the face lying on this surface

Conversion of elementary surfaces in the shape to surfaces of revolution

ShapeCustom::ConvertToRevolution()

```
TopoDS_Shape ShapeCustom::ConvertToRevolution(const TopoDS_Shape& S)
;
```

Purpose: Returns a new shape with all elementary periodic surfaces converted to Geom_SurfaceOfRevolution .

It uses the tool *ShapeCustom_ConvertToRevolution.*

Conversion of elementary surfaces in the shape to surfaces of revolution

ShapeCustom::ConvertToBSpline()

```
TopoDS_Shape ShapeCustom::ConvertToBSpline( const TopoDS_Shape& S,
        const Standard_Boolean extrMode,
        const Standard_Boolean revolMode,
        const Standard_Boolean offsetMode) ;
```

Purpose: Returns a new shape with all surfaces of linear extrusion, revolution and offset surfaces converted according to flags to Geom_BSplineSurface (with the same parameterization).

It uses the tool *ShapeCustom_ConvertToBSpline.*

Getting the history of modification of sub-shapes.

If, in addition to the resulting shape, you want to get the history of modification of sub-shapes you should not use the package methods described above and should use your own code instead:

1. Create a tool that is responsible for the necessary modification.
2. Create the tool BRepTools_Modifier that performs a specified modification in the shape.
3. To get the history and to keep the assembly structure use the method ShapeCustom::ApplyModifier.

Example

```
     //For scaling, the general calling syntax is:
     TopoDS_Shape scaled_shape = ShapeCustom::ScaleShape(shape, scale);
     //Note that scale is a real value.
     //You can refine your mapping process by using additional calls to
//follow shape mapping subshape by subshape. The following code,
//along with pertinent includes can be used:
     p_Trsf T;
     Standard_Real scale = 100; // for example!
     T.SetScale (gp_Pnt (0, 0, 0), scale);
     Handle(ShapeCustom_TrsfModification) TM = new
     ShapeCustom_TrsfModification(T);
     TopTools_DataMapOfShapeShape context;
     BRepTools_Modifier MD;
     TopoDS_Shape res = ShapeCustom::ApplyModifier (
     Shape, TM, context,MD );
     //The map, called context in our example, contains the history.
     //Substitutions are made one by one and all shapes are transformed.
     //To determine what happens to a particular subshape, you can use:
     TopoDS_Shape oneres = context.Find (oneshape);
     //In case there is a doubt, you can also add:
     if (context.IsBound(oneshape)) oneres = context.Find(oneshape);
     //You can also sweep the entire data map by using:
     TopTools_DataMapIteratorOfDataMapOfShapeShape
     //To do this, enter:
     for(TopTools_DataMapIteratorOfDataMapOfShapeShape
                   iter(context);iter(more ();iter.next ()) {
       TopoDs_Shape oneshape = iter.key ();
       TopoDs_Shape oneres = iter.value ();
     }
```

<u>Remove internal wires.</u>

Overview.

This tool is intended for removing internal wires whose contour area is less than the specified minimal area.

This tool is implemented as a new ShapeHealing operator in the package ShapeUpgrade.- **ShapeUpgrade_RemoveInternalWires**. This tool can work with various types of shapes:

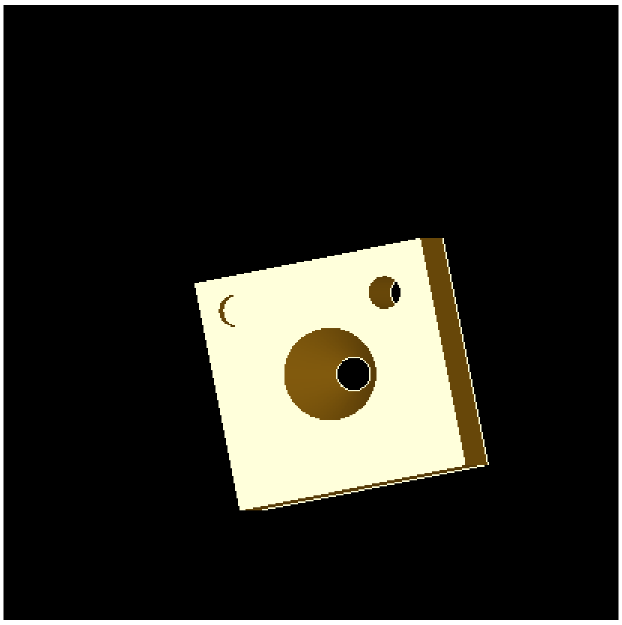
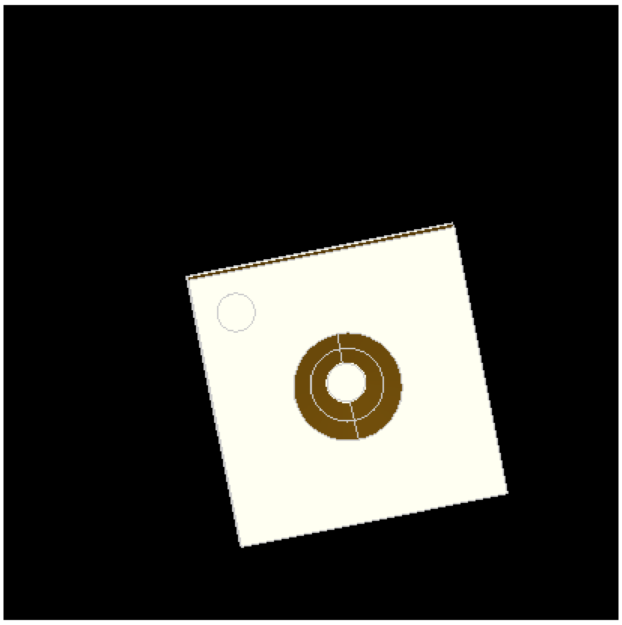
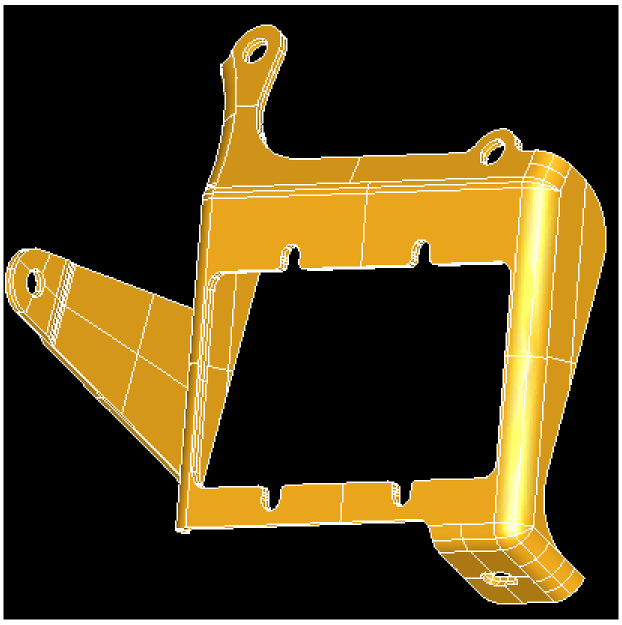
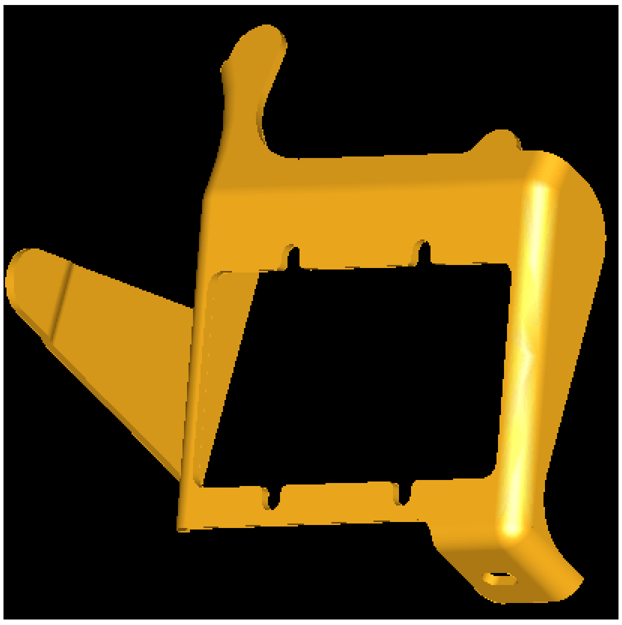
- Compound,
- Solid,
- Shell,
- Face

In the process of work this tool examines each face of the shape and removes all internal wires whose contour area is less than the minimal allowed area. If the flag RemoveFaceMode is set to TRUE, separate faces or a group of faces with outer wires consisting only of edges belonging to the removed internal wires are removed (seam edges are not taken into account).

Please note that such faces can be removed only for a sewed shape*.*

An example of using this tool is presented in the figures below:

Source shape **Result shape (RemoveFaceMode falg = TRUE)**

Please note that:

- **First case :**

  Three internal wires whose contour area is less than the specified minimal area have been removed.

  One internal face has been removed. The outer wire of this face consists of the edges belonging to the removed internal wires and a seam edge.

  Two other internal faces have not been removed because their outer wires consist not only of edges belonging to removed wires.

- **Second case:**

  Six internal wires whose contour area is less than the specified minimal area have been removed.

  Six internal faces have been removed. These faces can be united in groups of faces. Each group of faces has an outer wire consisting only of edges belonging to the removed internal wires. Such groups of faces are also removed.

Description of the implemented classes

A new class for removing internal wires has been created in the package **ShapeUpgrade**

New method for calculation of wire area in 3D space has been added to the package **ShapeAnalysis** – method ShapeAnalysis::ContourArea().

API class ShapeUpgrade_RemoveInternalWires

This class is inherited from the base class **ShapeUpgrade_Tool**.

- This class can be created with the help of an empty constructor or a constructor initialized by the shape.
- This class can be initialized by the shape with the help of the method Init() .
- The minimal allowed contour area in the 3D space should be specified by the method MinArea.
- Faces can be removed as described above by the flag RemoveFaceMode. The value of this flag can be set by the method RemoveFaceMode() (default value is equal to TRUE).

  Internal wires can be removed   by the methods Perform. Both methods Perform can not be carried out if the class has not been initialized by the shape. In such case the status of Perform is set to FAIL .

  The method Perform without arguments removes from all faces in the specified shape internal wires whose area is less than the minimal area.

  The other method Perform has a sequence of shapes as an argument. This sequence can contain faces or wires.

  If the sequence of shapes contains wires, only the internal wires are removed.

  If the sequence of shapes contains faces, only the internal wires from these faces are removed.

- The status of the performed operation can be obtained using the method Status()
- The resulting shape can be obtained using the method GetResult().

See the example:

//Initialixation class by shape.
```
Handle(ShapeUpgrade_RemoveInternalWires) aTool = new
ShapeUpgrade_RemoveInternalWires(inputShape);
```

//setting parameters
```
aTool->MinArea() = aMinArea;
aTool->RemoveFaceMode() = aModeRemoveFaces;
```

 //when method Perform is carried out on separate shapes.
```
aTool->Perform(aSeqShapes);
```

//when method Perform is carried out on whole shape.
```
aTool->Perform();
```

//check status set after method Perform
```
if(aTool->Status(ShapeExtend_FAIL) {
  cout<<”Operation failed”<< "<<"\n";
   return;
}

if(aTool->Status(ShapeExtend_DONE1)) {
    const TopTools_SequenceOfShape& aRemovedWires =aTool->RemovedWires();
     cout<<aRemovedWires.Length()<<" internal wires were removed"<<"\n";

  }


  if(aTool->Status(ShapeExtend_DONE2)) {
    const TopTools_SequenceOfShape& aRemovedFaces =aTool->RemovedFaces();
     cout<<aRemovedFaces.Length()<<" small faces were removed"<<"\n";

  }

   //getting result shape
  TopoDS_Shape res = aTool->GetResult();
```

Draw command

DRAW Command “**RemoveIntWires**” for the tool is implemented in the **class SWDRAW_ShapeUpgrade**.

The format of the command:

RemoveIntWires   result_shape min_area init_shape [faces or wires] [mode_removefaces]

Parameters of the command:

Mandatory parameters:

- result_shape – the resulting shape obtained through splitting.
- min_area – minimal allowed 3D contour area.
- intial_shape – the shape on which the splitting can be performed.

Optional parameters:

- **faces or wires –** sub-shapes belonging to the initial shape which should be processed.

  If faces exist, the internal wires will be removed only from the faces

  If wires exist, only the wires will be removed.

  By default the operation will be performed on all faces of the initial shape.

- **mode_removefaces**- can be set to 0 or 1. The Mode defines if the internal wires will be removed from separate faces or a group of faces whose outer wires consist only of edges. By default its value is set to TRUE. (the faces will be removed)

4.2.3. Tool for conversion of surfaces (Class ShapeCustom_Surface)

This class is intended

- to convert BSpline and Bezier surfaces to the analytical form (method C*onvertToAnalytical())*
- to make closed B-Spline surfaces to be periodic.(*method ConvertToPeriodic*)

To convert surfaces to analytical form this class analyzes the form and the closure of the source surface and defines whether it can be approximated by analytical surface of one of the following types:

Geom_Plane,

Geom_SphericalSurface,

Geom_CylindricalSurface,

Geom_ConicalSurface,

Geom_ToroidalSurface.

The conversion is done only if the new (analytical) surface does not deviate from the source one more than on the given precision.

Example

```
Handle(Geom_Surface) initSurf;

ShapeCustom_Surface ConvSurf(initSurf);

//conversion to analytical form

Handle(Geom_Surface) newSurf  =
ConvSurf.ConvertToAnalytical(allowedtol,Standard_False);

//or conversion to a periodic surface

Handle(Geom_Surface) newSurf  =
ConvSurf.ConvertToPeriodic(Standard_False);

//getting the maximum deviation of the new surface from the
//initial surface

Standard_Real maxdist = ConvSurf.Gap();
```

5. *Auxiliary tools for repairing, analysis and upgrading.*

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5.1. Tool for rebuilding shapes.

(ShapeBuild_ReShape)

This tool rebuilds a shape by making pre-defined substitutions on some of its components. During the first phase, it records requests to replace or remove some individual shapes. For each shape, the last given request is recorded. Requests may be applied as "Oriented" (i.e. only to an item with the same orientation) or not (the orientation of the replacing shape corresponds to that of the original one). Then these requests may be applied to any shape, which may contain one or more of these individual shapes.

This tool has a flag for taking the location of shapes into account (for keeping the structure of assemblies) (*ModeConsiderLocation).* If this mode is equal to Standard_True then shared shapes with locations will be kept. If this mode is equal to Standard_False then some different shapes will be produced from one shape with different locations after rebuilding. By default, this mode is equal to Standard_False.

To use this tool for the reconstruction of shapes it is necessary:

- Create this tool and use method *Apply()* for its initialization by the initial shape.

  Parameter <until> sets the level of shape type and requests are taken into account up to this level only. Sub-shapes of the type standing beyond the “line” set by parameter <until> will not be rebuilt and no further exploration will be done

- Replace or remove sub-shapes of the initial shape.

  Please NOTE:

  Each subshape can be replaced by a shape of the same type or by shape containing shapes of that type only (for example, TopoDS_Edge can be replaced by TopoDS_Edge, TopoDS_Wire or TopoDS_Compound containing TopoDS_Edges). If an incompatible shape type is encountered, it is ignored and flag FAIL1 is set in Status.

  For a subshape it is recommended to use method Apply before methods Replace and Remove because the subshape has already been changed for the moment by its previous modifications or modification of its subshape (for example TopoDS_Edge can be changed by a modification of its TopoDS_Vertex etc..).

- Use method *Apply* for the initial shape again to get the resulting shape after all modifications have been made

- Use method *Apply* to obtain the history of sub-shape modification.

Please NOTE:

In fact class ShapeBuild_ReShape is an alias for class BRepTools_ReShape. They differ only in queries of statuses in the ShapeBuild_ReShape class.

Example:

Using tool for getting result shape after modification of sub-shapes of initial shape

```
TopoDS_Shape initialShape…
//creation of a rebuilding tool
Handle(ShapeBuild_ReShape) Context = new ShapeBuild_ReShape.
```

```
//next step is optional. It can be used for keeping
//the assembly structure.
Context-> ModeConsiderLocation = Standard_True;

//initialization of this tool by the initial shape
Context->Apply(initialShape);
…
//getting the intermediate result for replacing subshape1 with
//the modified subshape1.
TopoDS_Shape tempshape1 = Context->Apply(subshape1);

//replacing the intermediate shape obtained from subshape1 with the
//newsubshape1.
Context->Replace(tempsubshape1,newsubshape1);
…
//for removing the subshape
TopoDS_Shape tempshape2 = Context->Apply(subshape2);
Context->Remove(tempsubshape2);

//getting the result and the history of modification
TopoDS_Shape resultShape = Context->Apply(initialShape);

//getting the resulting subshape from the subshape1 of the initial
//shape.
TopoDS_Shape result_subshape1 = Context->Apply(subshape1);
```

5.2. Enumeration for Status definition

(ShapExtend_Status)

Used to report the status after executing some methods that can either fail, do something, or do nothing. The status is a set of flags DONEi, FAILi, any combination of them can be set at the same time. For exploring the status, enumeration is used.

The values mean:

OK, -- Nothing is done, everything OK

DONE1, -- Something was done, case 1

…

DONE8, -- Something was done, case 8

DONE, -- Something was done (any of DONE#)

FAIL1, -- The method failed, case 1

…

FAIL8, -- The method failed, case 8

FAIL -- The method failed (any of FAIL# occurred)

5.3. Tool representing a wire

(Class ShapeExtend_WireData)

This tool provides a data structure necessary to work with the wire as with an ordered list of edges, and that is required for many algorithms. The advantage of this class is that it allows to work with incorrect wires.

The object of the class ShapeExtend_WireData can be initialized by TopoDS_Wire, and converted back to TopoDS_Wire.

An edge in the wire is defined by its rank number. Operations of accessing, adding and removing an edge at/to the given rank number are provided. Operations of circular permutation and reversing (both orientations of all edges and the order of edges) are provided on the whole wire as well.

This class also provides a method to check if the edge in the wire is a seam (if the wire lies on a face).

Example:

Removing edges from the wire and definition of whether an edge is a seam edge

```
TopoDS_Wire ini = ..
Handle(ShapeExtend_Wire) asewd = new ShapeExtend_Wire(initwire);
//Removing edge Edge1 from the wire.

Standard_Integer index_edge1 = asewd->Index(Edge1);
asewd.Remove(index_edge1);
//Definition of whether Edge2 is a seam edge
Standard_Integer index_edge2 = asewd->Index(Edge2);
asewd->IsSeam(index_edge2);
```

5.4. Tool for exploring the shapes

(Class ShapeExtend_Explorer)

This class is intended to explore shapes and convert different representations (list, sequence, compound) of complex shapes. It provides tools for:

- obtaining the type of the shapes in the context of TopoDS_Compound,
- exploring shapes in the context of TopoDS_Compound,
- converting different representations of shapes (list, sequence, compound).

5.5. Tool for attaching messages to objects

(Class ShapeExtend_MsgRegistrator)

This class attaches messages to objects (generic Transient or shape). The objects of this class are transmitted to the Shape Healing algorithms so that they could collect messages occurred during shape processing. Messages are added to the Maps (stored as a field) that can be used, for instance, by Data Exchange processors to attach those messages to initial file entities.

Example:

Sending message and getting message attached to object

```
Handle(ShapeExtend_MsgRegistrator) MessageReg = new
ShapeExtend_MsgRegistrator;
//attaches messages to an object (shape or entity)
Message_Msg msg..
TopoDS_Shape Shape1…
MessageReg->Send(Shape1,msg,Message_WARNING);
Handle(Standard_Transient) ent ..
MessageReg->Send(ent,msg,Message_WARNING);
//gets messages attached to shape
const ShapeExtend_DataMapOfShapeListOfMsg& msgmap =
                                          MessageReg->MapShape();
if (msgmap.IsBound (Shape1)) {
  const Message_ListOfMsg &msglist = msgmap.Find (Shape1);
  for (Message_ListIteratorOfListOfMsg iter (msglist);
                        iter.More(); iter.Next()) {
        Message_Msg msg = iter.Value();
  }
}
```

……

5.6. Tools for performance measurement

(Classes MoniTool_Timer and MoniTool_TimerSentry)

Timers are used for measuring the performance of a current operation or any part of code, and provide the necessary API. Timers are used for debugging and performance optimizing purposes.

6. *Shape Processing*

6.1. How you use Shape Processing.

The Shape Processing module allows defining and applying the general Shape Processing as a customizable sequence of Shape Healing operators. The customization is implemented via the user-editable resource file, which defines the sequence of operators to be executed and their parameters.

The Shape Processing functionality is implemented with the help of the XSAlgo interface. The main function XSAlgo_AlgoContainer::ProcessShape() does shape processing with specified tolerances and returns the resulting shape and associated information in the form of Transient.

Example of using this function:

```
TopoDS_Shape aShape = …;

Standard_Real Prec = …,

Standard_Real MaxTol = …;

TopoDS_Shape aResult;

Handle(Standard_Transient) info;

TopoDS_Shape aResult = XSAlgo::AlgoContainer()->ProcessShape(aShape,

       Prec, MaxTol., “Name of ResourceFile”, “NameSequence”, info
      );
```

You can use this possibility to create of your own sequence of operations and its management:

You have to create a resource file with the name <ResourceFile>, which includes the following string:

```
NameSequence.exec.op:     MyOper
```

<MyOper> - name of operation.

You should also input your own parameter for this operation in the resource file, for example:

```
NameSequence.MyOper.Tolerance: 0.01
```

(<Tolerance> - name of your parameter and 0.01 – its value)

You have to add the following string into the void ShapeProcess_OperLibrary::Init ():

```
ShapeProcess::RegisterOperator("MyOper",

             new ShapeProcess_UOperator(myfunction));
```

where < myfunction> is a function which implements the operation.

You have to create this function in the ShapeProcess_OperLibrary by the following way:

```
static Standard_Boolean myfunction (const

             Handle(ShapeProcess_Context)& context)

{

  Handle(ShapeProcess_ShapeContext) ctx =

             Handle(ShapeProcess_ShapeContext)::DownCast(context);

  if(ctx.IsNull()) return Standard_False;

  TopoDS_Shape aShape = ctx->Result();
```

```
     //receive our parameter:
     Standard_Real toler;
     ctx->GetReal("Tolerance", toler);
    // now you can make the necessary operations with <aShape>
    //using the received value of your parameter <Tolerance> from the
    //resource file  . . .


     return Standard_True;
  }
```

You can define some operations (with their parameters) <MyOper1>, <MyOper2>, <MyOper3> etc. and describe the corresponding functions in ShapeProcess_OperLibrary. After that you get a possibility to manage performing of the needed sequence using the specified name of operations and values of parameters in the resource file.

For example: if you input such string into the resource file

```
NameSequence.exec.op:    MyOper1,MyOper3
```

It means that the corresponding functions from ShapeProcess_OperLibrary will be performed with the original shape (<aShape>) using parameters defined for MyOper1 and MyOper3 in the resource file.

It is necessary to note that these operations will be performed step by step and the result obtained after performing the first operation will be used as the initial shape for the second operation.

6.2. Short description of existing operators.

Operator : DirectFaces

Description :

This operator sets all faces based on indirect surfaces, defined with left-handed coordinate systems as direct faces. This concerns surfaces defined by Axis Placement (Cylinders, etc). Such Axis Placement may be indirect, which is allowed in Cascade, but not allowed in some other systems. This operator reverses indirect placements and recomputes PCurves accordingly.

Operator : SameParameter

Indication :

This operator is required after calling some other operators, according to the computations they do. Its call is explicit, so each call can be removed according to the operators, which are either called or not afterwards. This mainly concerns splitting operators that can split edges.

Description :

This operator applies the computation SameParameter which ensures that various representations of each edge (its 3d curve, the pcurve on each of the faces on which it lies) give the same 3D point for the same parameter, within a given tolerance.
For each edge coded as "same parameter", deviation of curve representation is computed and if the edge tolerance is less than that deviation, the tolerance is increased so that it satisfies the deviation. No geometry modification, only an increase of tolerance is possible.
For each edge coded as "not same parameter" the deviation is computed like in

the first case. Then an attempt is made to achieve the edge equality to "same parameter" by means of modification of 2d curves. If the deviation of this modified edge is less than the original deviation then this edge is returned, otherwise the original edge (with non-modified 2d curves) is returned with an increased (if necessary) tolerance.
Computation is done by call to the standard CAS.CADE algorithm BRepLib::SameParameter.

Parameters :

Boolean : Force (default = false) (optional)

If True, encodes all edges as "not same parameter" then runs the computation. Else, the computation is done only for those edges already coded as "not same parameter".

Real : Tolerance3d (optional)

If not defined, the local tolerance of each edge is taken for its own computation. Else, this parameter gives the global tolerance for the whole shape.

Operator : BSplineRestriction

Description :

Used for conversion of surfaces, curves 2d curves to BSpline surfaces with a specified degree and a specified number of spans.

This operator performs approximations on surfaces, curves and 2d curves with a specified degree, max number of segments, 2d tolerance, 3d tolerance. The specified continuity can be reduced if the approximation with a specified continuity was not done successfully.

Parameters :

Boolean : SurfaceMode

to consider the surfaces

Boolean : Curve3dMode

to consider the 3d curves

Boolean : Curve2dMode

to consider the 2d curves

Real : Tolerance3d

3d tolerance to be used in computation

Real : Tolerance2d

2d tolerance to be used when computing 2d curves

GeomAbs_Shape (C0 G1 C1 G2 C2 CN) : Continuity3d

continuity required in 2d

GeomAbs_Shape (C0 G1 C1 G2 C2 CN) : Continuity2d

continuity required in 3d

Integer : RequiredDegree
Integer : RequiredNbSegments

Boolean : PreferDegree

in case of a conflict, parameter RequiredDegree , by default, is set as priority, else RequiredNbSegments has a priority

Boolean : RationalToPolynomial

This parameter serves for conversion of BSplines to polynomial form

Integer : MaxDegree

the maximum allowed Degree, if RequiredDegree cannot be reached

Integer : MaxNbSegments

the maximum allowed NbSegments, if RequiredNbSegments cannot be reached

The following flags allow to manage the conversion of special types of curves or surfaces, in addition to BSpline. They are controlled by SurfaceMode Curve3dMode Curve2dMode respectively. By default, only BSplines and Bezier Geometries are considered.

Boolean : OffsetSurfaceMode
Boolean : LinearExtrusionMode
Boolean : RevolutionMode
Boolean : OffsetCurve3dMode
Boolean : OffsetCurve2dMode

Boolean : PlaneMode

Boolean : BezierMode

Boolean : ConvCurve3dMode

Boolean : ConvCurve2dMode

For each of the Mode parameters listed above, if it is True, the specified geometry is converted to BSpline, otherwise only its basic geometry is checked and converted (if necessary) keeping the original type of geometry (revolution, offset, etc).

Boolean :SegmentSurfaceMode

Has effect only for Bsplines and Bezier surfaces. When False a surface will be replaced by a Trimmed Surface, else new geometry will be created by splitting the original Bspline or Bezier surface.

Operator : ElementaryToRevolution

Description :

This operator converts elementary periodic surfaces to SurfaceOfRevolution.

Operator : SplitAngle

Description :

This operator splits surfaces of revolution, cylindrical, toroidal, conical, spherical surfaces in the given shape so that each resulting segment covers not more than the defined number of degrees.

Parameters :

Real : Angle

the maximum allowed angle for resulting faces

Real : MaxTolerance

maximum tolerance used in computations

Operator : SurfaceToBSpline

Description :

This Operator converts some specific types of Surfaces, to BSpline (according to parameters).

Parameters :

Boolean : LinearExtrusionMode

to convert surfaces of Linear Extrusion

Boolean : RevolutionMode

to convert surfaces of Revolution

Boolean : OffsetMode

to convert Offset Surfaces

OpFerator : ToBezier

Indication :

This operator is used for data supported as Bezier only.

Description :

This Operator converts various types of geometries to Bezier

Parameters :

Boolean : SurfaceMode
Boolean : Curve3dMode
Boolean : Curve2dMode
Real : MaxTolerance

used in computation of conversion

Boolean : SegmentSurfaceMode (default is True)

Has effect only for Bsplines and Bezier surfaces. When False a surface will be replaced by a Trimmed Surface, else new geometry will be created by splitting the original Bspline or Bezier surface.

The following parameters are controlled by one of the above SurfaceMode or Curve3dMode or Curve2dMode (according to the case):

Boolean : Line3dMode
Boolean : Circle3dMode
Boolean : Conic3dMode

Boolean : PlaneMode
Boolean : RevolutionMode
Boolean : ExtrusionMode
Boolean : BSplineMode

Operator : SplitContinuity

Description :

This operator splits a shape in order to have each geometry (surface, curve 3d, curve 2d) correspond the given criterion of continuity.

Parameters :

Real : Tolerance3d
Integer (GeomAbs_Shape) : CurveContinuity
Integer (GeomAbs_Shape) : SurfaceContinuity
Real : MaxTolerance

Warning :

Because of algorithmic limitations in the operator BSplineRestriction (in some particular cases, this operator can produce unexpected C0 geometry), if SplitContinuity is called, it is recommended to call it after BSplineRestriction.

Continuity Values will be set as GeomAbs_Shape (i.e. C0 G1 C1 G2 C2 CN) besides direct integer values (resp. 0 1 2 3 4 5).

Operator : SplitClosedFaces

Description :

This operator splits faces, which are closed even if they are not revolutionary or cylindrical, conical, spherical, toroidal. This corresponds to BSpline or Bezier surfaces which can be closed (whether periodic or not), hence they have a seam edge.
As a result, no more seam edges remain. The number of points allows to control the minimum count of faces to be produced per input closed face.

Parameters :

Integer : NbSplitPoints

number of points to use for splitting (the number of intervals produced is then NbSplitPoints+1)

Real : CloseTolerance

tolerance used to determine if a face is closed

Real : MaxTolerance

used in the computation of splitting

Operator : FixGaps

Description :

This operator must be called when FixFaceSize and/or DropSmallEdges are called. Using Surface Healing may require an additional call to BSplineRestriction to ensure that modified geometries meet the requirements for BSpline.

This operators repairs geometries which contain gaps :

gaps between edges in wires (always performed) gaps on faces, controlled by parameter SurfaceMode

Gaps on Faces are fixed by using algorithms of Surface Healing

Parameters :

Real : Tolerance3d

Tolerance to reach in 3d. If a gap is less than this value, it is not fixed.

Boolean : SurfaceMode

Mode of fixing gaps between edges and faces (yes/no)

Integer : SurfaceAddSpans

Number of spans to add to the surface in order to fix gaps

GeomAbs_Shape (C0 G1 C1 G2 C2 CN) : SurfaceContinuity

Minimal continuity of a resulting surface

Integer : NbIterations

Number of iterations

Real : Beta

Elasticity coefficient for modifying a surface [1-1000]

Reals : Coeff1 to Coeff6

Energy coefficients for modifying a surface [0-10000]

Real : MaxDeflection

Maximal deflection of surface from an old position.

Warning :

This operator may change the original geometry. In addition, it is CPU consuming, and it may fail in some cases.
ALSO : FixGaps can help only when there are gaps obtained as a result of removal of small edges that can be removed by DropSmallEdges (or FixFaceSize).

Operator : FixFaceSize

Description :

This operator removes faces, which are:

small in all directions (spot face)
small in one direction (strip face)

Parameters :

Real : Tolerance

The minimal dimension, which is used to consider a face, is small enough to be removed.

Operator : DropSmallEdges

Description :

This operator drops edges in a wire, and merges them with adjacent edges, when they are smaller than the given value (Tolerance3d) and when the topology allows such merging (i.e. same adjacent faces for each of the merged edges)

Free (non-shared by adjacent faces) small edges can be also removed in case if they share the same vertex Parameters.

Parameters :

Real : Tolerance3d

Dimension used to determine if an edge is small.

Operator : FixShape

Description :

This operator may be added for fixing invalid shapes. It performs various checks and fixes, according to the modes listed hereafter. Management of a set of fixes can be performed by flags as follows:

if the flag for a fixing tool is set to 0 , it is not performed
if set to 1 , it is performed in any case
if not set, or set to -1 , for each shape to be applied on, a check is done to evaluate whether a fix is needed. The fix is performed if the check is positive

By default, they are "not set", i.e. evaluated for each individual shape.

Parameters :

Real : Tolerance3d

basic tolerance used for fixing

Real : MaxTolerance3d

maximum allowed value for the resulting tolerance

Real : MinTolerance3d

minimum allowed value for the resulting tolerance.

Boolean : FixFreeShellMode
Boolean : FixFreeFaceMode
Boolean : FixFreeWireMode
Boolean : FixSameParameterMode
Boolean : FixSolidMode
Boolean : FixShellMode
Boolean : FixFaceMode
Boolean : FixWireMode
Boolean : FixOrientationMode
Boolean : FixMissingSeamMode
Boolean : FixSmallAreaWireMode

Boolean (not checked) : ModifyTopologyMode

Specifies the mode for modifying topology. Should be False (default) for shapes with shells and can be True for free faces.

Boolean (not checked) : ModifyGeometryMode

Specifies the mode for modifying geometry. Should be False if geometry is to be kept and True if it can be modified.

Boolean (not checked) : ClosedWireMode

Specifies the mode for wires. Should be True for wires on faces and False for free wires.

Boolean (not checked) : PreferencePCurveMode (not used)

Specifies the preference of 3d or 2d representations for an edge

Boolean : FixReorderMode
Boolean : FixSmallMode
Boolean : FixConnectedMode
Boolean : FixEdgeCurvesMode
Boolean : FixDegeneratedMode
Boolean : FixLackingMode
Boolean : FixSelfIntersectionMode
Boolean : FixGaps3dMode
Boolean : FixGaps2dMode
Boolean : FixReversed2dMode
Boolean : FixRemovePCurveMode
Boolean : FixRemoveCurve3dMode
Boolean : FixAddPCurveMode
Boolean : FixAddCurve3dMode
Boolean : FixSeamMode
Boolean : FixShiftedMode
Boolean : FixEdgeSameParameterMode
Boolean : FixSelfIntersectingEdgeMode
Boolean : FixIntersectingEdgesMode
Boolean : FixNonAdjacentIntersectingEdgesMode

Operator : SplitClosedEdges

Indication :

This operator handles closed edges i.e. edges with one vertex. Such edges are not supported in some receiving systems.

Description :

This operator splits topologically closed edges (i.e. edges having one vertex) into two edges. Degenerated edges and edges with a size of less than Tolerance are not processed.

Parameters :

None.

7. *Messaging mechanism*

7.1. Overview

During the process of shape fixing or upgrading various messages about modification, warnings and fails can be are generated. The messaging mechanism allows you to generate messages, which will be sent to the target medium of your choice. This can be either a file or your screen. The messages may be failure ones and/or warning messages providing information on events such as analysis, fixing or upgrading of shapes.

The messaging mechanism is provided by the package Message. These packages are intended for message management and run-time monitoring and provide the following tools for run-time work with messages:

- managing files containing messages, (*Message_MsgFile);*
- managing messages and filling them with parameters, (*Message_Msg*);
- managing trace files.(*Message_TraceFile*);

The API enumeration is the following:

Message_Gravity

7.2. Enumeration Message_Gravity

This enumeration is used for defining message gravity.

Definition

```
enum Message_Gravity {
       Message_FAIL,
       Message_WARNING,
       Message_INFO
};
```

Purpose: Message_FAIL: the message is fail, Message_WARNING: the message is a warning, Message_INFO: the message is information.

7.3. Tool for loading a message file into memory

(Class Message_MessageFile)

You define your messages using the class Message_MsgFile. This class allows you to load your message file into memory. Note that you need to create your message file before loading it into memory, as its path will be used as the argument to load it. Each message in the message file is identified by its key. The user can get the text content of the message by specifying the message key.

Format of the message file

The message file is an ASCII file, which defines a set of messages. Each line of the file must have a length of less than 255 characters.

All the lines in the file starting with the exclamation sign (perhaps preceded by spaces and/or tabs) are considered as comments and are ignored.

A message file may contain several messages. Each message is identified by its key (string).

Each line in the file starting with the dot character "." (perhaps preceded by spaces and/or tabs) defines the key. The key is a string starting with a symbol placed after the dot and ending with the symbol preceding the ending of the newline character "\n".

All the lines in the file after the key and before the next keyword (and which are not comments) define the message for that key. If the message consists of several lines, the message string will contain newline symbols "\n" between each line (but not at the end).

The following example illustrates the structure of a message file:

```
!This is a sample message file

!------------------------------

!Messages for ShapeAnalysis package

!

.SampleKeyword

Your message string goes here

!

!...

!

!End of message file
```

Loading the message file

You load this sample file into memory using the method Message_MsgFile::LoadFile, taking as an argument the path to your file as in the example below:

```
Standard_CString MsgFilePath = "(path)/sample.file";

Message_MsgFile::LoadFile (MsgFilePath);
```

7.4. Tool for managing filling messages

(Class Message_Msg)

The class Message_Msg allows you to use the message file, which you have loaded as a template. This class provides a tool for preparing the message, filling it with parameters, storing and outputting to the default trace file.

A message is created from a key: this key identifies the message to be created in the message file. The text of the message is taken from the loaded message file (class Message_MsgFile is used).

The text of the message can contain places for parameters, which are to be filled by the proper values when the message is prepared. These parameters can be of the following types:

string - coded in the text as "%s",

integer - coded in the text as "%d",

real - coded in the text as "%f".

The user fills out the parameter fields in the text of the message by calling the corresponding methods. Both the original text of the message and the input text with substituted parameters are stored in the object.

You can use the methods AddInteger, AddReal and AddString to fill out the message parameters.

The prepared and filled message can be output to the default trace file. The text of the message (either original or filled) can be obtained also.

Example

```
!This is a sample message file

!------------------------------
```

```
!Messages for ShapeAnalysis package
.SampleKeyword
Found %d edges in the %s file.
...
!End of message file
```

Example

```
Message_Msg msg01 ("SampleKeyword");
//Creates the message msg01, identified
//in the file by the keyword SampleKeyword
msg1.AddInteger (73);
msg1.AddString ("SampleFile");
//fills out the code areas
```

7.5. Tool for managing trace files

(Class Message_TraceFile)

This class is intended to manage the trace file (or stream) for outputting messages and current trace level. Trace level is an integer number, which is used when messages are sent. Generally, 0 means minimum, > 0 various levels. If the current trace level is lower than the level of the message it is not output to the trace file. The trace level is to be managed and used by the users.

There are two ways of using trace files:

define an object of Message_TraceFile, with its own definition (file name or cout, trace level), and use it where it is defined,

use the default trace file (file name or cout, trace level), usable from anywhere.

Use the constructor method to define the target file and the level of the messages as in the example below:

```
Message_TraceFile myTF
        (tracelevel, “tracefile.log”, Standard_False);
```

<tracelevel> is a Standard_Integer and modifies the level of messages. It has the following values and semantics:

- 0: gives general information such as the start and end of process
- 1: gives exceptions raised and fail messages
- 2: gives the same information as 1 plus warning messages.

<filename> is the string containing the path to the log file.

The Boolean set to False will rewrite the existing file. When set to True, new messages will be appended to the existing file.

Creating a new log file

You can also create a new default log file using the method SetDefault with the same arguments as in the constructor.

Changing the trace level

You can change the default trace level by using the method SetDefLevel. In this way, you modify the information received in the log file.

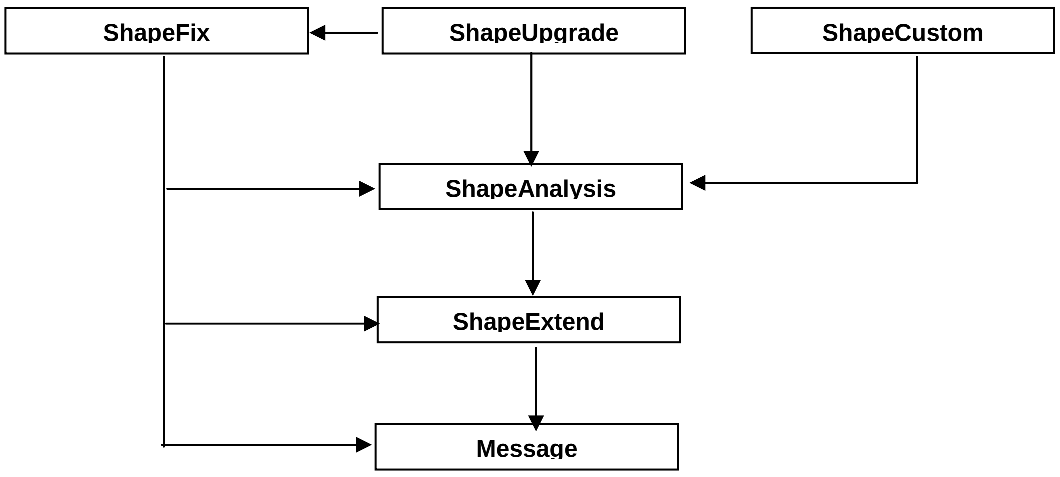
Choosing the screen output

You can close the log file and set the default trace output to the screen display instead of the log file using the method SetDefault without any arguments.

8. *Appendix A*

8.1. Dependencies of API packages

The following diagram shows dependencies of API packages:

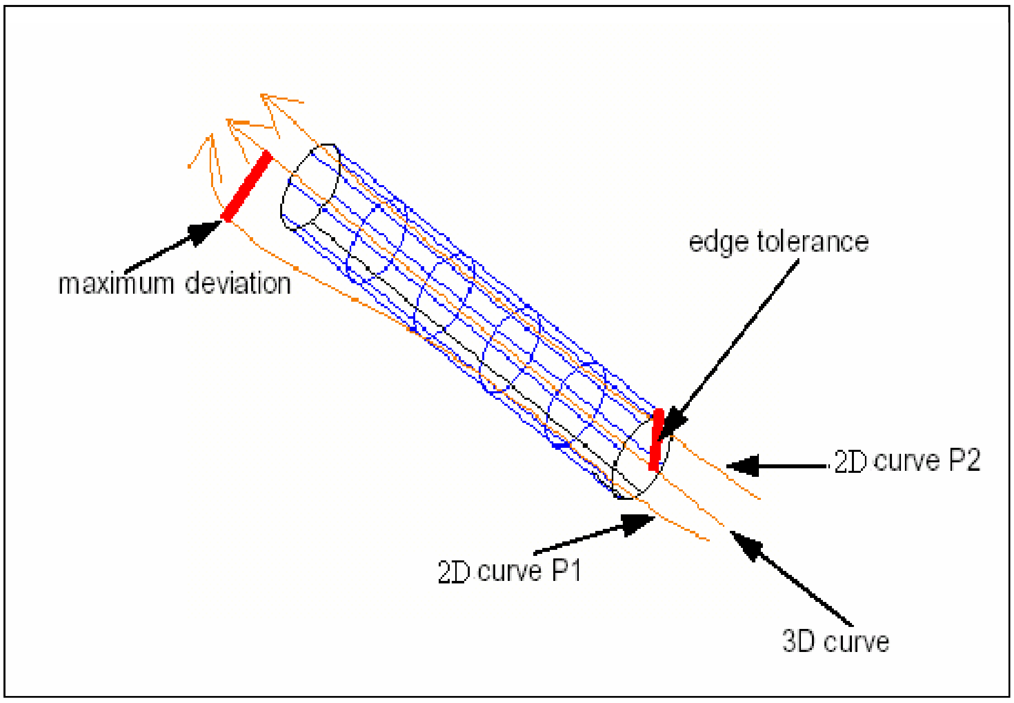


9. *Appendix B*

9.1. Examples of use

9.1.1. ShapeAnalysis_Edge and ShapeFix_Edge

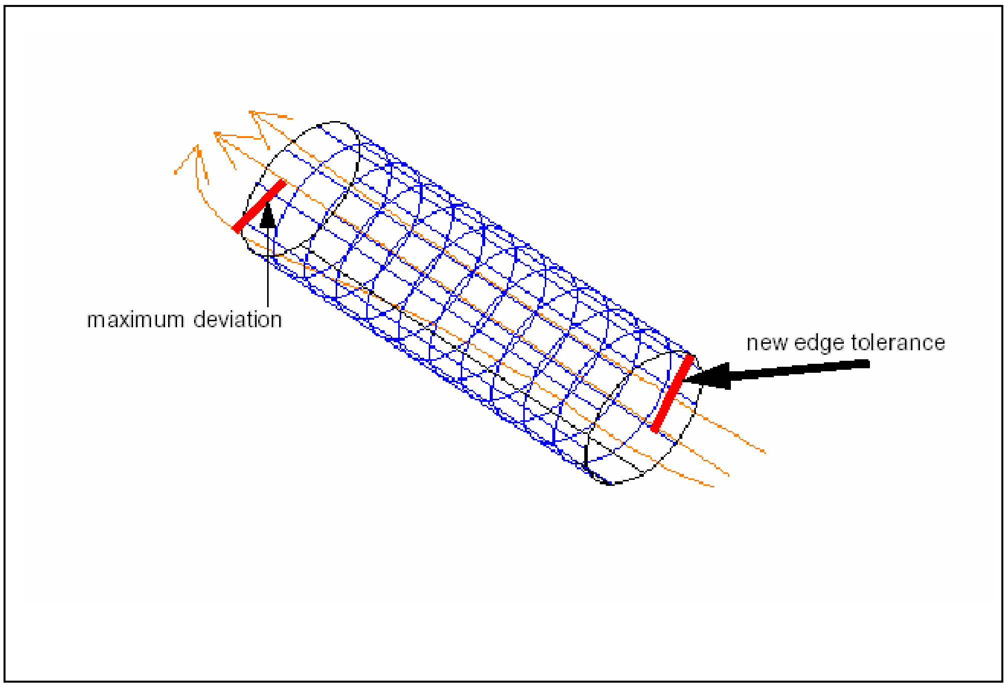
In this example an edge is shown where the maximum deviation between the 3D curve and 2D curve P1 is greater than the edge tolerance.



Example

```
ShapeAnalysis_Edge sae;
TopoDS_Face face = ...;
TopoDS_Wire wire = ...;
Standard_Real precision = 1e-04;
ShapeFix_Edge sfe;
Standard_Real maxdev;
if (sae.CheckSameParameter (edge, maxdev)) {
  cout<<“Incorrect SameParameter flag”<<endl;
  cout<<“Maximum deviation “<<maxdev<< “, tolerance “
           <<BRep_Tool::Tolerance(edge)<<endl;
  // Maximum deviation between pcurve and
  // 3D curve is greater than tolerance
  sfe.FixSameParameter();
  cout<<“New tolerance “<<BRep_Tool::Tolerance(edge)<<endl;
  // Tolerance is increased to englobe the deviation
}
```

The result is an increased edge tolerance:



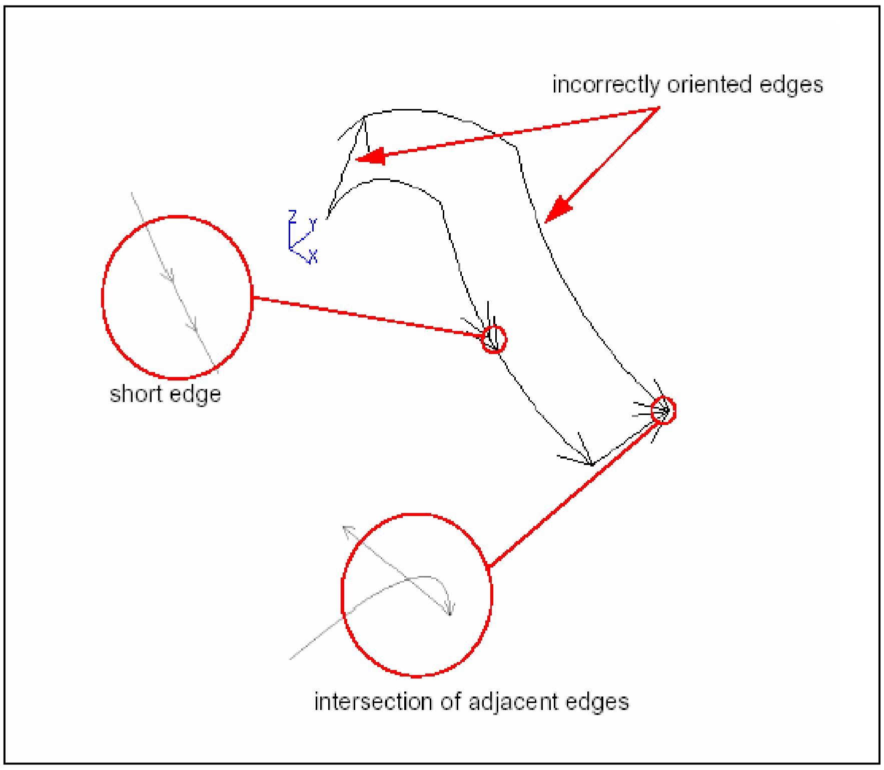
9.1.2. ShapeAnalysis_Wire and ShapeFix_Wire

This wire is first analyzed to check that:

• the edges are correctly oriented

• there are no edges that are too short and

• that there are no intersecting adjacent edges.

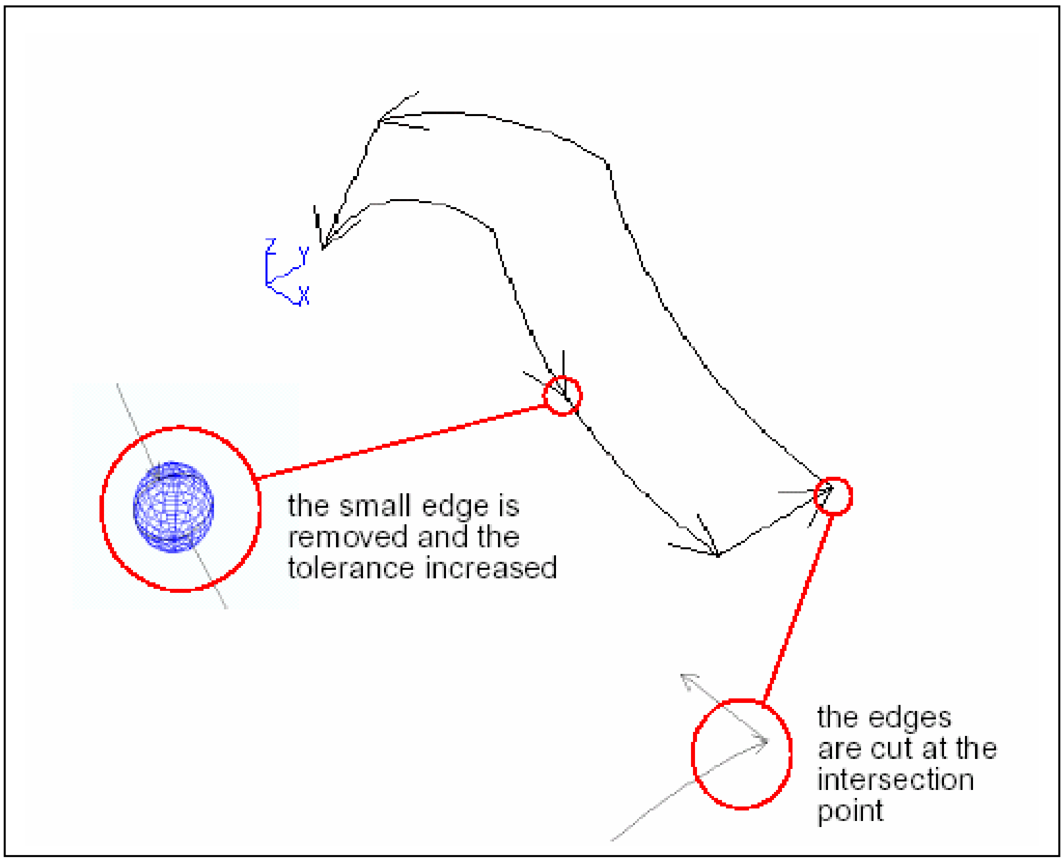


Example

```
TopoDS_Face face = ...;
TopoDS_Wire wire = ...;
Standard_Real precision = 1e-04;
ShapeAnalysis_Wire saw (wire, face, precision);
ShapeFix_Wire sfw (wire, face, precision);
if (saw.CheckOrder()) {
  cout<<“Some edges in the wire need to be reordered”<<endl;
  // Two edges are incorrectly oriented
  sfw.FixReorder();
  cout<<“Reordering is done”<<endl;
}
// their orientation is corrected
if (saw.CheckSmall (precision)) {
  cout<<“Wire contains edge(s) shorter than “<<precision<<endl;
```

```
          // An edge that is shorter than the given
          // tolerance is found
          Standard_Boolean LockVertex = Standard_True;
          if (sfw.FixSmall (LockVertex, precision)) {
            cout<<“Edges shorter than “<<precision<<“ have been removed”
                               <<endl;
            //The edge is removed
          }
        }
        if (saw.CheckSelfIntersection()) {
          cout<<“Wire has self-intersecting or intersecting
                               adjacent edges”<<endl;
          // Two intersecting adjacent edges are found
          if (sfw.FixSelfIntersection()) {
            cout<<“Wire was slightly self-intersecting. Repaired”<<endl;
            // The edges are cut at the intersection point so
            // that they no longer intersect
          }
        }
```

Here is the result:


Z
Y
X
the small edge is removed and the tolerance increased
the edges are cut at the intersection point

9.1.3. MoniTool_Timer

Using timers in XSDRAWIGES.cxx and IGESBRep_Reader.cxx for analysis performance command "igesbrep":

```
XSDRAWIGES.cxx
  ...
  #include <MoniTool_Timer.hxx>
  #include <MoniTool_TimerSentry.hxx>
  ...
  MoniTool_Timer::ClearTimers();
  ...
  MoniTool_TimerSentry MTS("IGES_LoadFile");
  Standard_Integer status = Reader.LoadFile(fnom.ToCString());
  MTS.Stop();
  ...
  MoniTool_Timer::DumpTimers(cout);
  return;
```

```
IGESBRep_Reader.cxx
  ...
  #include <MoniTool_TimerSentry.hxx>
  ...
  Standard_Integer nb = theModel->NbEntities();
  ...
  for (Standard_Integer i=1; i<=nb; i++) {
    MoniTool_TimerSentry MTS("IGESToBRep_Transfer");
    ...
    try {
      TP.Transfer(ent);
      shape = TransferBRep::ShapeResult (theProc,ent);
    }
    ...
  }
```

Result

DumpTimer() after translation file:

TIMER: IGES_LoadFile	Elapsed: 1.0 sec CPU User: 0.9 sec CPU Sys: 0.0 sec hits: 1
TIMER: IGESToBRep_Transfer	Elapsed: 14.5 sec CPU User: 4.4 sec CPU Sys: 0.1 sec hits: 1311