Appendix A.
Reference Summary

Topic: Ignore

This appendix provides a summary of the reference items defined in the Component Name. Just the name and a brief description of each item is given. There is a separate section for each type of item (class, function, Scheme extension, test harness command, etc.).

Classes

Topic: Ignore

ATTRIB_EFINT	Defines an attribute to record the intersections of an edge of one body with a face of the other body, during a Boolean operation.
ATTRIB_FACEINT	Defines an attribute to record the intersection of a face of one body with a face of the other body during a Boolean operation.
ATTRIB_INTCOED	Defines an attribute for linking intersection graph entities with the relevant body entities.
ATTRIB_INTEDGE	Defines an attribute for linking intersection edges with the intersecting entities.
ATTRIB_INTGRAPH	Defines an attribute for classifying shells and lumps of two bodies participating in a Boolean operation.
ATTRIB_INTVERT	Defines an attribute for linking graph vertices with the intersection record(s) giving rise to them.
glue_options	Class to hold information and options for a glue operation.
NO_MERGE_ATTRIB	Specifies a user-defined attribute that signals that the edge is not to be merged out of the body.

shell_lump . . . . . Records the classification of shells or wires that do not contribute to any intersection.

Enumerations

Topic: Ignore

BOOL_TYPE . . . . . Specifies the type of Boolean operation.

face_body_rel . . . . . Specifies the relationship between face and body entities.

NDBOOL_KEEP . . . . . For a non–destructive Boolean operation, this optional flag may be used to specify the preservation of either or both input bodies. If the blank body is to be preserved, the result body will be returned via the BODY pointer, result_body.

Functions

Topic: Ignore

api_boolean . . . . . Executes a general Boolean operation.

api_boolean_chop_body . . . . . Executes Boolean intersect and subtract operations on two bodies.

api_boolean_chop_complete . . . . . Completes the last steps Boolean intersect and subtract operations on two bodies.

api_boolean_complete . . . . . Finishes a Boolean operation.

api_boolean_glue . . . . . Executes a specialized Boolean operation, where the intersection graph is known to lie along a set of coincident faces.

api_boolean_start . . . . . Starts a Boolean operation.

api_bool_make_intersection_graph . . . . . Computes all the steps to return the intersection graph between two bodies. Do not remove the attributes attached to the entities.

api_check_entity_ff_ints . . . . . Checks all faces for improper intersections.

api_check_list_ff_ints . . . . . Checks all faces for improper intersections.

api_clean_body	Removes all edges (faces and associated data) that are not necessary to support the topology of the body.
api_clean_entity	Removes all edges and associated data that are not needed to support the topology of the entity.
api_clean_wire	Removes the attributes and extra coedges present on a wire body generated by the section or slice operation.
api_complete_intersection_graph	Determines the intersection graph between two bodies. Do not remove the attributes attached to the entities.
api_convert_to_spline	Converts an entity from analytic to spline.
api_detect_short_edges	Detects edges whose lengths are less than the tolerance given and replaces the edges with TVERTEXes.
api_detect_sliver_faces	Returns all 2-edge and 3-edge sliver faces from a body whose maximum distance among the edges is smaller than the given tolerance.
api_fafa_int	Determines the intersection between two faces.
api_fixup_intersection	Fix up intersection entities created by api_update_intersection().
api_imprint	Intersects two bodies and imprints their intersection graph on both without otherwise changing them.
api_imprint_complete	Finishes an imprint operation.
api_imprint_stitch	Combines bodies along their face-face intersection curves and at coincident vertices.
api_imprint_stitch_complete	Imprints bodies and then stitches them along the face-face intersection curves.
api_initialize_booleans	Initializes the Boolean library.
api_intersect	Executes a Boolean intersect operation on two bodies.
api_join_edges	Joins a list of edges into one single edge.

api_merge_faces . . . . . Removes all faces of a specified geometry type if they are not necessary to define the body.

api_planar_slice . . . . . Slices a BODY with a plane.

api_refresh_entity_pattern . . . . . Refreshes the elements of a pattern to incorporate changes made to one of them.

api_regularise_entity . . . . . Removes all faces, edges and vertices (and associated data) that are not necessary to support the topology of the entity.

api_remove_face . . . . . Removes a face from a body.

api_remove_no_merge_attrib . . . . . Removes a NO_MERGE_ATTRIB to each edge in the input list of edges.

api_remove_wire_edge . . . . . Removes a wire edge from a body and creates a new wire body from it.

api_replace_edge_with_tvertex . . . . . Replaces an edge or list of edges with a tolerant vertex.

api_replace_face_with_tedge . . . . . Replaces a 2 or 3-edge face with a tolerant edge.

api_selectively_imprint . . . . . Imprints a subset of the faces of the tool body with a subset of the faces of the blank body.

api_selectively_intersect . . . . . Intersects an array of faces of one body with an array of faces of another body.

api_set_no_merge_attrib . . . . . Sets a NO_MERGE_ATTRIB to each edge in the input list of edges.

api_slice . . . . . Determines the intersection graph between two bodies.

api_slice_complete . . . . . Finishes a slice operation.

api_slice_of_model . . . . . Creates a new model by a slice based on a clipped copy of the model.

api_split_edges_at_poles . . . . . Splits the edges of an entity at the poles.

api_split_face . . . . . Splits a face along a specified *u* or *v* isoparameter curve.

api_split_face_at_g_disc . . . . . Splits a face along ”u” or ”v” isoparametric lines at G1 or G2 discontinuities.

api_split_periodic_faces . . . . . Splits all periodic faces (along *u*, *v*, or both) to ensure that they are well formed.

api_stitch . . . . . Stitches faces along edges and vertices of identical geometry.

api_subtract . . . . . Executes a Boolean subtract operation.

api_terminate_booleans . . . . . Terminates the Boolean library.

api_uncover_face . . . . . Removes the surface of a face, leaving its edges.

api_unhook_face . . . . . Removes a face from a body.

api_unhook_wire_edge . . . . . Removes a wire edge from a body, placing wire in returned body.

api_unite . . . . . Executes a Boolean unite operation.

api_unite_wires . . . . . Unites the wires of the tool body with the wires of the blank.

api_unstitch_nonmani . . . . . Decomposes an input body along its nonmanifold vertices and edges.

api_update_intersection . . . . . Creates a surf_surf_int intersection structure to be used in place of an actual intersection.

is_ATTRIB_EFINT . . . . . Determines if an ENTITY is an ATTRIB_EFINT.

is_ATTRIB_FACEINT . . . . . Determines if an ENTITY is an ATTRIB_FACEINT.

is_ATTRIB_INTCOED . . . . . Determines if an ENTITY is an ATTRIB_INTCOED.

is_ATTRIB_INTEDGE . . . . . Determines if an ENTITY is an ATTRIB_INTEDGE.

is_ATTRIB_INTGRAPH . . . . . Determines if an ENTITY is an ATTRIB_INTGRAPH.

is_ATTRIB_INTVERT . . . . . Determines if an ENTITY is an ATTRIB_INTVERT.

is_NO_MERGE_ATTRIB	Determines if an ENTITY is a NO_MERGE_ATTRIB.

Options

Topic: Ignore

all_free_edges	Determines whether all edges on coincident faces are processed as free edges.
check_ee_int_always	Controls when edge/edge intersections are performed during face/face checking.
check_ff_int	Sets additional validity checking of the body.
check_ff_tangent_int	Sets additional validity checking of the body.
keep_second_edge	Sets whether or not the second edge will be kept for merging.
keep_second_face	Sets whether or not the second face will be kept for merging.
merge	Sets merging of common geometry.
merge_spline_vertex	Sets merging of spline edges connected by a two–edge vertex.
new_periodic_splitting	Controls how the periodic face splitting algorithm is used.
slow_bool4	Determines whether to regenerate pcurves.
split_face_checking	Controls whether or not split faces are checked for small faces.
subsetting	Sets the level of surface subsetting when trimming.
trim_faces	Sets the trimming of splines to the parameter bounds of the face.

Scheme Extensions

Topic: Ignore

bool:chop	Simultaneously finds the intersection and difference between two bodies.

bool:clip	Creates a copy of a model clipped to two parallel planes.
bool:glue–subtract	Performs a subtraction on two bodies that have a set of overlapping, coincident faces and no penetrating face–face intersections.
bool:glue–subtract–inter–graph	Performs a subtraction on two bodies that have a set of overlapping, coincident faces and no penetrating face-face intersections.
bool:glue–unite	Unites two bodies that have a set of overlapping, coincident faces and no penetrating face–face intersections.
bool:glue–unite–inter–graph	Unites two bodies that have a set of overlapping, coincident faces and no penetrating face-face intersections.
bool:intersect	Intersects two or more bodies.
bool:join–edges	Joins multiple edges into a single edge.
bool:merge	Combines faces and edges of equivalent geometry.
bool:merge–faces	Combines specific faces on a body.
bool:nonreg–chop	Simultaneously finds the nonregularized intersection and difference between two bodies.
bool:nonreg–intersect	Intersects two or more nonregularized bodies.
bool:nonreg–subtract	Subtracts one or more nonregularized bodies from a body.
bool:nonreg–unite	Unites two or more nonregularized bodies.
bool:regularise	Regularizes an entity.
bool:sel–imprint	Imprints the intersection graph of a set of selected faces of the tool body and a set of selected faces of blank body.
bool:subtract	Subtracts one or more bodies from a body.
bool:trim–faces	Trims the surfaces of the given faces.

bool:unite . . . . . Unites two or more bodies.

bool:unite–wires . . . . . Unites the wires of the tool body with the wires of the blank body.

bool:wifa–imp . . . . . Imprints a wire on one or more faces of another body.

edge:set–no–merge–attrib . . . . . Sets a NO_MERGE_ATTRIB to each edge in the input list of edges.

entity:refresh–pattern . . . . . Refreshes the elements of a pattern to incorporate changes made to one of them.

entity:remove–pcurves . . . . . Removes pcurves from all coedges on analytic faces of the body or the selected entity.

entity:reset–pcurves . . . . . Removes and then adds back pcurves from all coedges of the body or the selected entity.

entity:spline–convert . . . . . Creates a new entity which is a spline equivalent of the original entity.

face:intersect . . . . . Gets the intersection curve between two faces.

face:remove . . . . . Removes a face from a body.

face:split . . . . . Splits a face along “u” or “v” at a given value.

face:split–at–disc . . . . . Splits a face along G1 or G2 discontinuities.

face:uncover . . . . . Removes the surface of a face, leaving its edges.

face:unhook . . . . . Removes a face from a body.

glue:options . . . . . Sets the options in the data structure to be used by glue operations.

solid:check–ff–intersections . . . . . Checks all faces for improper intersections.

solid:imprint . . . . . Imprints curves of intersection of two bodies onto the faces of bodies.

solid:imprint–stitch . . . . . Joins body1 and body2 along the intersection graph.

solid:inter–graph . . . . . Gets the intersection graph between two bodies and returns it as a wire body.

solid:intersect	Intersects a list of solids.
solid:planar–slice	Slices a solid body with a plane to produce a wire body.
solid:slice	Gets the intersection graph between two bodies and returns it as a wire body.
solid:split	Splits all periodic faces of a body along the seams.
solid:stitch	Joins two bodies (faces) along edges or vertices of identical geometry.
solid:subtract	Subtracts a list of solids from a solid.
solid:unhook–wire–edge	Unhooks an edge belonging to a wire from a body and returns a new wire-body.
solid:unite	Unites two or more solids.
solid:unstitch–nm	Decomposes the input body along nonmanifold edges and vertices.
tolerant:detect–short–edges	Returns all edges from either a body or a wire that are shorter in length than the specified tolerance.
tolerant:detect–sliver–faces	Returns all sliver faces from a body whose maximum distance among the edges is smaller than the given tolerance.
tolerant:replace–edge–with–tvertex	Replaces a list of edges with tolerant vertices.
tolerant:replace–face–with–tedge	Replaces a 2– or 3-edge face with a tolerant edge.
wire:clean	Removes the attributes and extra coedges present on a wire body generated by the section or slice operation.