

CAM Express Academic Bundle – NXCAD100

This product is a complete set of CAM functions. It will address the NC Programming needs of a wide variety of shops.

UG11210 *TURNING*

UG11214 ***POST EXEC*** - the executable portion of the NX machine tool postprocessor, which converts internal NX CAM tool paths into machine specific output. This is an essential module for all NC programmers that require the capability to postprocess NC output.

UG11216 ***POST BUILDER*** - allows users to interactively create NC postprocessor programs that in turn can be executed using NX Post Exec to produce machine specific output. Standard milling, drilling, and turning machines are supported. It contains Windows like structures with drag and drop options for defining the different postprocessor parameters. In addition, it provides the capability to manually modify its output so that, users with TCL programming experience, can extend postprocessing features not available in NX Post Builder.

UG11220 *CAM VISUALIZE*

UG11221 ***FACET MACHINING*** - allows the user to machine facet based models such as STL or Parasolid facet models that do not have any associated Geometry. This module will be used with reverse engineering or scanned models.

UG11245 *VARIABLE AXIS MILLING*

UG11250 *GRAPH TOOL PATH EDITOR*

UG11260 *CORE AND CAVITY MILLING*

UG11265 ***FLOW CUT*** - Enables customers to generate tool paths along concave corners and valleys formed by Part Surfaces. The processor automatically determines the direction and order of the flow cuts using certain rules based on machining best practices. The resulting Tool Path is optimized in such a way that the tool remains in contact with the part as much as possible and minimizes Non-Cutting moves. Manual assembly is available for the cases where human intervention can improve the results. Flow Cut is available only for Fixed Contour operations.

UG11270 *SEQUENTIAL MILLING*

UG11275 *NURBS PATH GENERATOR*

UG11280 ***WIRE EDM*** - allows for the wire EDM cutting of parts in 2 through 4 axis mode. Wireframe as well as solid models are supported to define the part to be cut. Many types of wire operations are supported such as multi-pass profiling, wire reversing and area removal.

UG11284 ***SHOP DOC*** - module designed to provide NC program documentation capabilities for NX CAM.

UG11290 ***FEATURE BASED AUTOMATION*** - Feature Based Machining provides automation tools for machining holes, pockets, and faces with pre-defined processes.

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UG11580 *ISV - SIMULATION & Advanced Simulation* - Provides an out-of-the-box capability to do machine simulation and collision detection using a set of pre-defined generic machine tools.

UG13085 *ELECTRO-MECHANICAL ROUTING*

UG14010 *CONTOUR PROFILING*

UG2715 *CAM BASE*

UG2735 *PLANAR MILLING*

