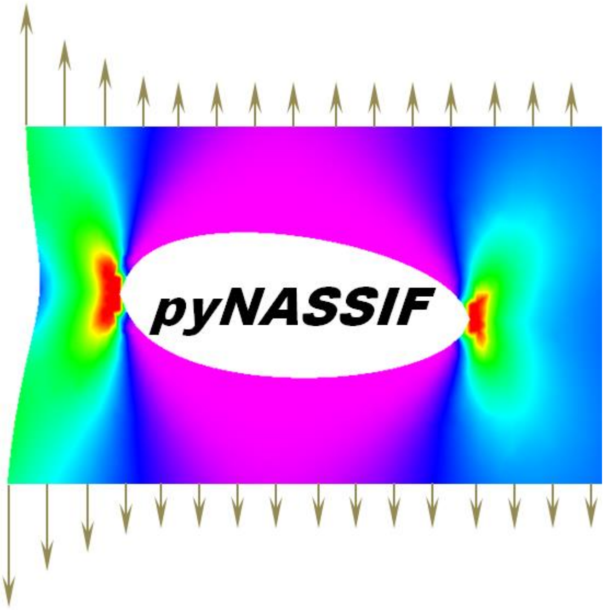
Stehlin Engineering	Technical Memo	SE-TM-14-001
	T. Stehlin	10-Jan-2014, Issue 1 Page 1 of 27

pyNASSIF
Theory and Manual



SUMMARY/CONCLUSIONS

This technical memo presents the pyNASSIF (python NASTRAN Stress Intensity Factor) tool, which aim is to compute the stress intensity factor (K) and the stress intensity factor correction (β) along a given crack path in function of the crack length in structures idealized by PLATE elements in NASTRAN FEM models. It is able to deal with two crack tips at the same time (1 thru crack or 2 edge cracks).

pyNASSF is a script written in python, which interacts with the NASTRAN solver and the NASTRAN input/output files to retrieves all information relevant for the calculation of K and β.

Description of the program is provided in this document together with the background theory.

Validation of the tool is also presented by comparing β solutions calculated by pyNASSIF with the values from other sources like AFGROW, NASGRO and FRANC2D for typical cracks configurations.

DESCRIPTION OF CHANGES

Issue No	Section of Page affected	Description of Changes
1	All	Initial Issue

TOOLS

Tool Name	Version	Date
pyNASSIF.py	1.0.8	05.01.2014

UNITS

pyNASSIF is set up to work with the SI modified [mm] system of units, i.e. length in [mm], force in [N], stress in [MPa] and stress intensity in [MPa√mm] . Note that for consistency with AFGROW input format stress intensities in [MPa√m] are also computed by pyNASSIF.

Note that other system of unit may also be used but consistency between load, length, stress and energy should be kept in any case.

Table of Content

1 INTRODUCTION

The aim of pyNASSIF tool is to compute the stress intensity factor (K) and the stress intensity factor correction (β) along a given crack path in function of the crack length in structures idealized by PLATE elements in NASTRAN FEM models.

pyNASSIF is based on the elastic-energy release rate concept. This method uses relationships that exist between the K and the elastic-energy content (U) of the cracked structure. The elastic-energy content is calculated using FEM model of the cracked structure and an interface with NASTRAN is provided to automatize all analyses steps. Background equations for this method are presented in Section 3.

This method permits to compute K values and β solutions for edge crack(s) or thru crack for geometries, which are difficult to handle using standard solutions available in standard CG software like AFGROW and NASGRO or by compounding of solutions of other sources or when conservatism in the analysis has to be reduced. This also permits to assess the impact of parameters like fasteners flexibility, stiffener effect, secondary bending effects, etc. on the crack growth behaviour. When required, more complex structural behaviour such as non-linearity due to large displacements or contacts can also be assessed.

Validation of the method is presented in Section 4 by comparing the results obtained by pyNASSIF and other sources for simple cases (edge crack in a plate under tension for 2 different boundary conditions, eccentric hole in plate under tension, eccentric hole in plate with bearing loading, edge crack approaching a thickness change, centered thru crack in a plate under tension and eccentric thru crack in a plate under tension). This validation exercise shows that pyNASSIF provides results very close to all other sources.

Section 5 presents the program limitations as well as some recommendations to help the analyst to make appropriate use of the tool.

2 PROGRAM DESCRIPTION

The pyNASSIF program is written in python version 3.3. The program runs NASTRAN analyses for predefined crack steps, extracts the external work for each crack step and computes K and β in function of the edge crack length. Calculation of K and β depends on parameters like material properties, cracked plate thickness and length of crack steps. The program is able to handle up to 2 crack tips, i.e. one edge crack, two edge cracks or an internal thru crack. The program automatically retrieves information relevant for the K and β calculations from the NASTRAN input file (nodes positions along crack path to derive crack increment and crack length, plate elements thicknesses and E-modulus) and from the NASTRAN F06 output file (external work or strain energy).

2.1 FEM MODEL

Before to start a pyNASSIF analysis a FEM model dedicated to pyNASSIF has to be built. This model is a NASTRAN FEM model containing the structure to be calculated and the crack(s) path(s) along which the K and β will be calculated. Each crack is closed with RBE2 elements connecting the crack fronts. The crack is implemented in the FEM by performing the following steps:

1. Defining the crack path for a chosen load case. Typically the load case chosen for the analysis is the reference load case used to normalize the spectrum and the crack path is perpendicular to the maximum principal stress vector for mode I or parallel to the maximum shear stress vector for modes II & III.
2. Refining the mesh in this zone.
3. Unzip the elements along the crack(s) path(s).
4. Close the crack with RBE2 elements numbered in ascending order along the crack path.

2.2 INPUTS

The master file is a comma separated format file, which contains the necessary information to run the script. The solutions available are presented in Table 2-1 and the keywords used in the master file are presented in Table 2-2. Comments can be implemented in the master file by starting the line with "$" (same as NASTRAN). A typical master file is presented in Figure 2-1.

The FEM file is a standard NASTRAN input file.

The crack files are simply NASTRAN files containing the RBE2 elements defining the crack (one file per crack tip). Note that the RBE2 elements shall be numbered in ascending order along the crack path.

Table 2-1 : Solutions implemented in pyNASSIF

CrkType	Description
110	One crack
120	Two crack tips , run crack tip 1 first (all steps) and then crack tip 2 (all steps)
121	Two cracks, run crack tip 1 and tip 2 alternatively (crack 1 step i, then crack 2 step i, etc.)
122	Two cracks, for each crack tip 1 step run all steps of crack tip 2 (β-factors matrices)

Table 2-2 : Keywords used in pyNASSIF Master File

Keyword	Type	Description	Used for
DFEM	string	NASTRAN input file	analysis
Crk01File	string	NASTRAN file with RBE2 elements defining the crack tip 1	analysis
Crk02File	string	NASTRAN file with RBE2 elements defining the crack tip 2	analysis
CrkType	integer	Solution type, see Table 2-1	analysis
a01	real	Initial crack size of tip 1	analysis
Crk01End	integer	Final node on crack tip 1 path	analysis
a02	real	Initial crack size of tip 2	analysis
Crk02End	integer	Final node on crack tip 2 path	analysis
LCref	string	Name of the analysis load case (reference load case)	info
fref	real	Reference stress. This stress is used for β-factors calculation	analysis
NasExe	string	Path to the NASTRAN solver	analysis
RunNastran	string	"Yes" or "No". Define if NASTRAN solver has to be run. Useful when NASTRAN results files are already available.	analysis
Energy	string	"Strain" or "ExtWork". Define which method is used for energy extraction. Both methods provide very similar results. The advantage of the "Strain" method is that models with contacts and non-linear runs are supported.	analysis

```
$ NASTRAN files of structure and crack(s)
DFEM,Baseline.dat
Crk01File,Crack01.dat
Crk01File,Crack02.dat
$ Crack type and definitions (initial size and end Node)
CrkType,110
a01,1.9
Crk01End,10005
a02,1.8
Crk01End,10105
$ Reference conditions
LCref,Tension Load Case (100MPa)
fref,100
$ Path to NASTRAN solver
NasExe,C:/FEMAPv111/nastran/bin/nastran.exe
$ Nastran analysis to be performed: "Yes" or "No"
RunNastran,Yes
$ Energy Extraction: "Strain" or "ExtWork"
Energy,ExtWork
```

Figure 2-1: Typical Master File

2.3 OUTPUT

The output is a comma separated file, which can be opened in excel for subsequent processing. The first 9 lines of the output are dedicated to general information for ease of traceability such as pyNASSIF version, pyNASSIF solution number, NASTRAN files, initial crack(s) size(s) and reference load case and stress, see Figure 2-2. The second part of the ouput file depends on the pyNASSIF solution used:

1. For solutions 110, 120 and 121 the output will be for each crack tip a block of 1 line per crack increment containing FEM information applicable to the crack step (which RBE2 was disconnected, node and position of the crack tip, crack increment, crack length, E-modulus and thickness at the crack tip, energy, energy increment, K and β), see Figure 2-3.
2. For solutions 122 the output will be two matrixes of β solutions, one for each crack tip, with β values for each crack length of one tip depending on the crack length of the other tip, see Figure 2-4.

pyNASSIF Version	1.0.8	
DFEM Name	Baseline.dat	
Crack01 File	Crack01.dat	
Crack02 File	Crack02.dat	
Crack Type	122	
Crack01 a0	1.667	[mm]
Crack02 a0	1.667	[mm]
Reference LC Name	Tension 100MPa	
Reference Stress	100	[MPa]

Figure 2-2: First Part of Output File

Crack01																			
CrkFile-ID	RBE2-ID	Node-ID	CS-ID	Xnd	Ynd	Znd	QUAD4-ID	Prop-ID	E	t	Da	a	a	EW	dEW	K	K	K0	Beta
[-]	[-]	[-]	[-]	[mm]	[mm]	[mm]	[-]	[-]	[MPa]	[mm]	[mm]	[mm]	[m]	[KJ]	[KJ]	[MPaSQR(mm)]	[MPaSQR(m)]	[MPaSQR(m)]	[-]
cr01-100	0	10001	0	-250	0	0					0	0	0	145833.33					
cr01-101	10001	10002	0	-248	0	0	1128	1	72000	1	2	1	0.001	145834.06	0.73	162.111073	5.126402247	5.604991216	0.914613788
cr01-102	10002	10003	0	-246	0	0	1127	1	72000	1	2	3	0.003	145836.89	2.83	319.1864659	10.0935623	9.708129563	1.03970206
cr01-103	10003	10004	0	-244	0	0	1089	1	72000	1	2	5	0.005	145841.94	5.05	426.3801121	13.48332303	12.53314137	1.075813528
cr01-104	10004	10005	0	-242	0	0	1086	1	72000	1	2	7	0.007	145849.11	7.17	508.0551151	16.06611341	14.82941286	1.083395112
cr01-105	10005	10006	0	-240	0	0	1087	1	72000	1	2	9	0.009	145858.39	9.28	577.9965398	18.27785545	16.81497365	1.086998757
cr01-106	10006	10007	0	-238	0	0	1088	1	72000	1	2	11	0.011	145869.91	11.52	643.9875775	20.3646753	18.58965282	1.095484434
cr01-107	10007	10008	0	-236	0	0	1096	1	72000	1	2	13	0.013	145883.44	13.53	697.9111691	22.06988899	20.20908323	1.092077693
cr01-108	10008	10009	0	-234	0	0	1097	1	72000	1	2	15	0.015	145898.88	15.44	745.5467792	23.57625925	21.70803764	1.086061285
cr01-109	10009	10010	0	-232	0	0	1099	1	72000	1	2	17	0.017	145916.08	17.2	786.8926229	24.88372962	23.10997082	1.076752966
cr01-110	10010	10011	0	-230	0	0	1098	1	72000	1	2	19	0.019	145935.77	19.69	841.9263626	26.62404928	24.43159029	1.089738693
cr01-111	10011	10012	0	-228	0	0	1178	1	72000	1	2	21	0.021	145957.3	21.53	880.3862789	27.84025862	25.68529652	1.083898665
cr01-112	10012	10013	0	-226	0	0	1177	1	72000	1	2	23	0.023	145980.48	23.18	913.4987685	28.88736748	26.88059356	1.074655119
cr01-113	10013	10014	0	-224	0	0	1176	1	72000	1	2	25	0.025	146005.86	25.38	955.8660994	30.22714012	28.02495608	1.078579393
cr01-114	10014	10015	0	-222	0	0	1166	1	72000	1	2	27	0.027	146032.61	26.75	981.325634	31.0322413	29.12438869	1.065507044
cr01-115	10015	10016	0	-220	0	0	1165	1	72000	1	2	29	0.029	146061.19	28.58	1014.337222	32.07615937	30.18380144	1.062694486
cr01-116	10016	10017	0	-218	0	0	1164	1	72000	1	2	31	0.031	146091.48	30.29	1044.241351	33.02181097	31.20727035	1.058144804
cr01-117	10017	10018	0	-216	0	0	1163	1	72000	1	2	33	0.033	146123.73	32.25	1077.4971	34.07345007	32.19822318	1.058240074

Figure 2-3: Second Part of Output File 1 for solutions 110, 120 and 121

Beta-01 Matrix										
a2/a1	1.60737464	2.67944704	3.74642325	4.81569402	5.88524674	6.95601438	8.02562035	9.12456235	10.2536061	11.3826826
1.64132537	1.13825958	1.07180388	1.07490507	1.04626772	1.02478362	0.95624404	0.94619005	0.90199959	0.88176985	0.86836711
2.7330561	1.35245391	1.21260767	1.13466691	1.10542484	1.05507932	1.00116232	0.97035198	0.9228981	0.90733369	0.89152763
3.82965485	1.58589278	1.33868262	1.2322959	1.16157307	1.10372574	1.03569526	1.00166167	0.95004795	0.93219676	0.91410152
4.92746591	1.8103022	1.46958345	1.33515894	1.21512959	1.15941032	1.06911334	1.03947344	0.97644319	0.96237271	0.93613122
6.02040897	2.009809	1.61838837	1.41906081	1.29622552	1.20384818	1.11736804	1.07595724	1.0084662	0.98584867	0.96295989
7.11700771	2.19122577	1.74147518	1.49827159	1.36323281	1.25509849	1.15604267	1.11124386	1.03950316	1.00877845	0.9890611
8.21360646	2.35873028	1.85641878	1.59434187	1.41814992	1.30433661	1.20080917	1.14544395	1.06963993	1.03672774	1.00945626
9.30533719	2.49990657	1.96464897	1.67511856	1.47964601	1.35952911	1.23687752	1.1786521	1.09315131	1.06394306	1.02944743
10.4019359	2.64795601	2.06722047	1.75217533	1.54693664	1.40511251	1.27881742	1.21094992	1.12184782	1.08522403	1.05390327
11.4985347	2.78815515	2.18605957	1.82598319	1.60350119	1.4492629	1.31942487	1.2424084	1.14982837	1.11125229	1.07780434

Figure 2-4: Second Part of Output File 1 for solution 122, Beta Matrix on Tip 1

3 THEORY

According to Ref. [1], Chapter 11.2.3.2, the stress intensity at the crack tip of a cracked structure can be computed indirectly via the strain energy calculation of the loaded cracked structure. This method uses the relationships that exist between the stress intensity factor (K) and the elastic-energy content (U) of the cracked structure.

The stress-intensity factor K is computed as follows:

$$K = \sqrt{\frac{dU}{da}\frac{\overline{E}}{t}}$$

with: t = plate thickness

$\overline{E}$ = E in plane stress Young's modulus

$\overline{E}$ = $\frac{E}{(1-\nu^2)}$ in plane strain Young's modulus (not implemented in current pyNASSIF version)

da = crack length increment

dU = Energy released during growth of da

The elastic energy content and the compliance of cracked structures are obtained for a range of crack sizes. Differentiation with respect to crack size gives K from the above equations. The advantage of the elastic-energy content and compliance methods is that an extremely fine mesh is not necessary, since accuracy of crack-tip stresses is not required. A disadvantage is that differentiation procedures can introduce errors.

Accordingly, the stress intensity factor correction (β) can be computed as follows:

$$\beta = \frac{K}{K_0} = \frac{\sqrt{\frac{dU}{da}\frac{\overline{E}}{t}}}{\sigma\sqrt{\pi \cdot a}}$$

pyNASSIF uses the difference between the external work or strain energy calculated by NASTRAN for two subsequent crack sizes (a, a+da) to compute dU.

4 Validation

pyNASSIF results are compared to theoretical curves from the literature, Ref [2], and to results obtained by other programs such as AFGROW [3] & [4], NASGRO [5] and FRANC2D [6].

4.1 1 TIP

In this section the solutions 110 and 120 described in Table 2-1 are verified, for which 1 crack tip at a time is propagated.

4.1.1 Edge Crack in a Finite Width Plate: Uniaxial Tensile Stress without Bending Constraints

Plate is 500mm wide and 1 mm thick and loaded by a constant stress of 100 MPa, see Figure 4-1. As shown in Figure 4-2 and Figure 4-3, comparison with [2] to [6] show that β solution calculated by pyNASSIF is in very good agreement with the other sources.

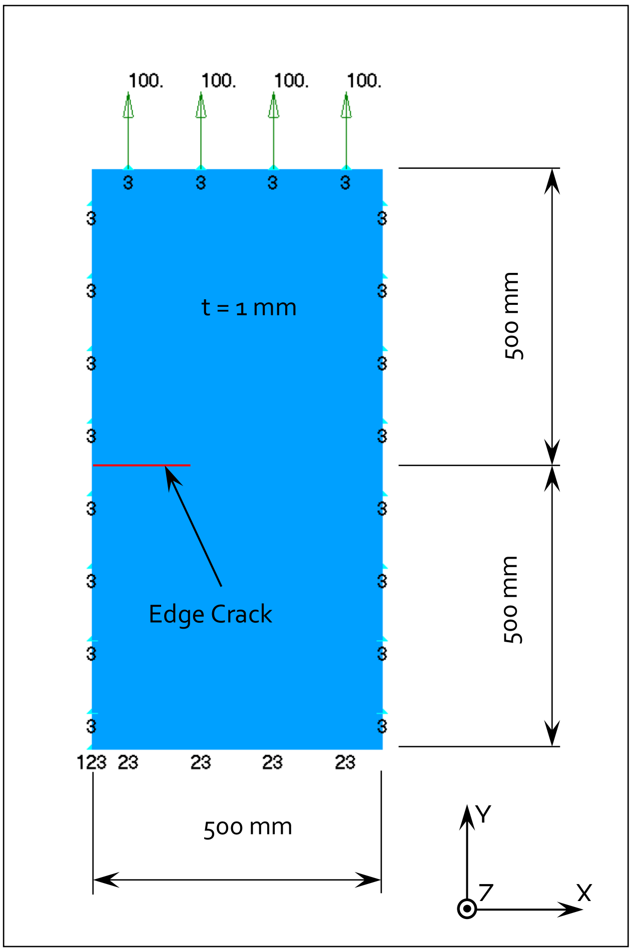


Figure 4-1: Geometry, Loads and Boundary Conditions

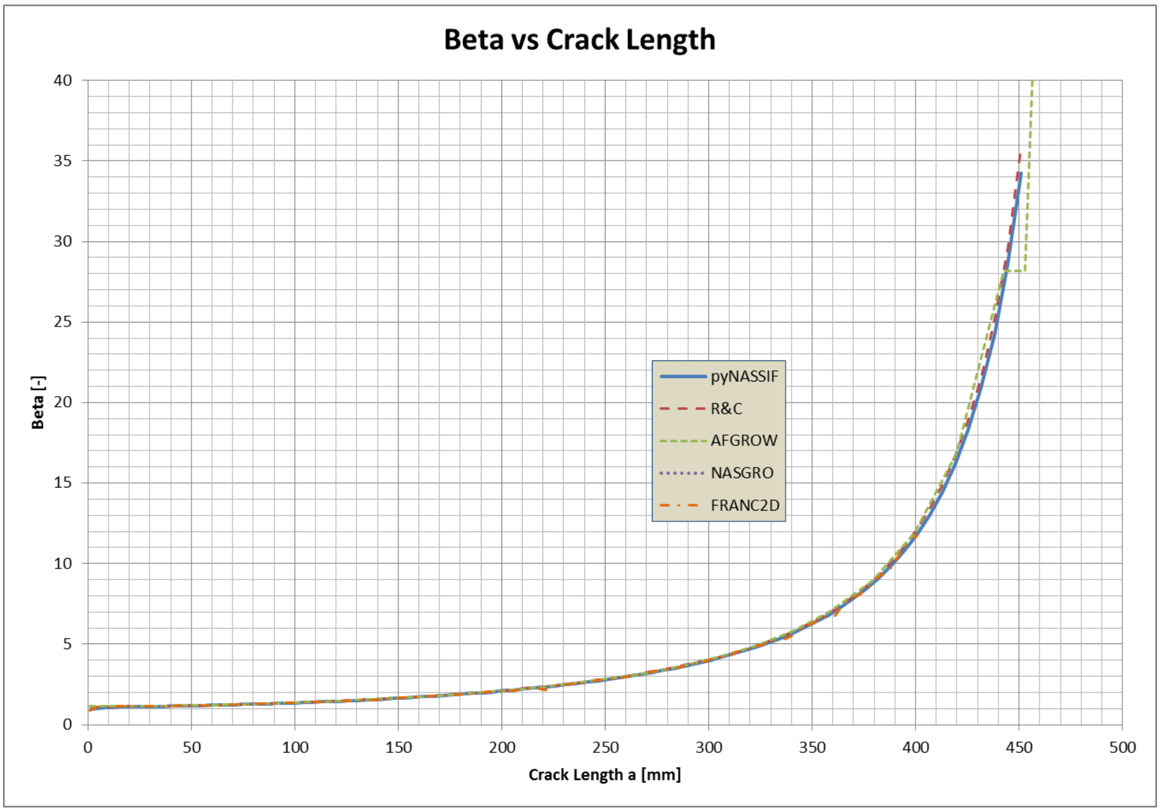


Figure 4-2: β Solution for Edge Crack in a Plate under Tension, 0 mm < a < 450 mm

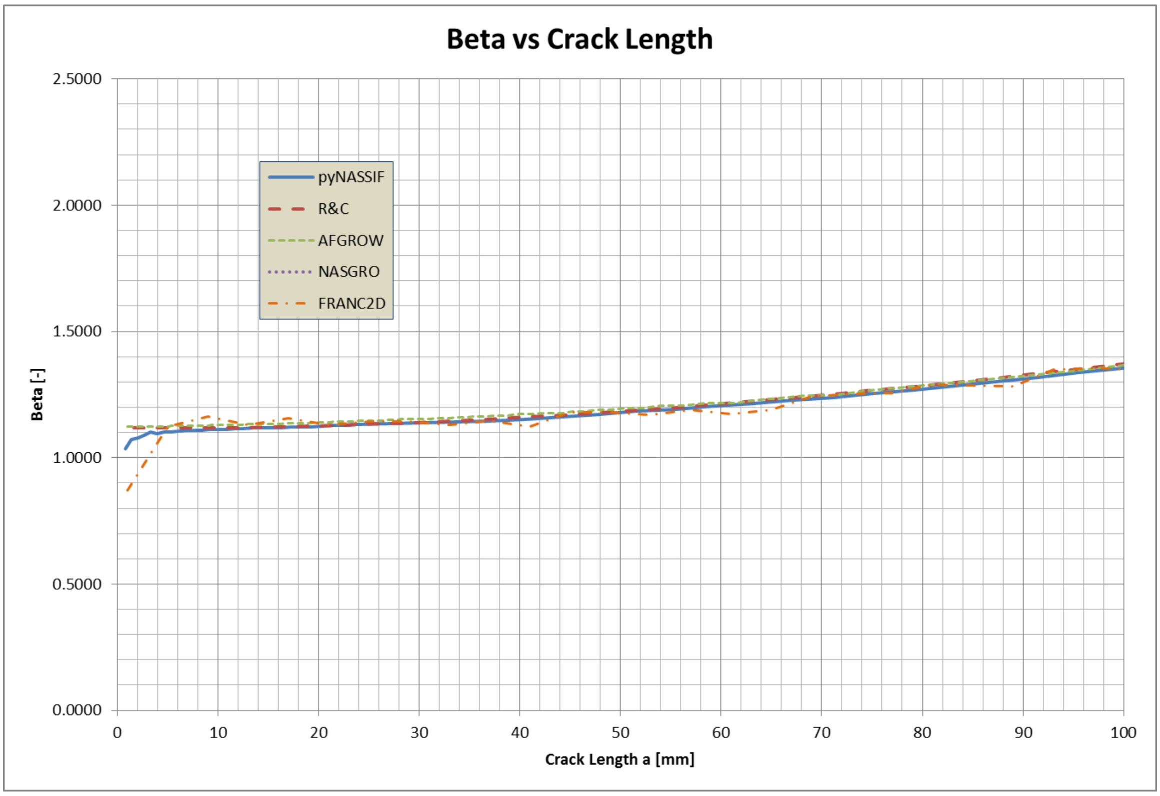


Figure 4-3: β Solution for Edge Crack in a Plate under Tension, 0 mm < a < 100 mm

4.1.2 Edge Crack in a Finite Width Plate: Uniaxial Tensile Stress with Guided End

This case is selected because it can be easily implemented in FRANC2D. Plate is 500mm wide and 1 mm thick and loaded by a constant stress of 200 MPa, see Figure 4-4. As shown in Figure 4-5, FRANC2D results show fluctuation around pyNASSIF results. It is concluded that β solution calculated by pyNASSIF is in very good agreement with FRANC2D results.

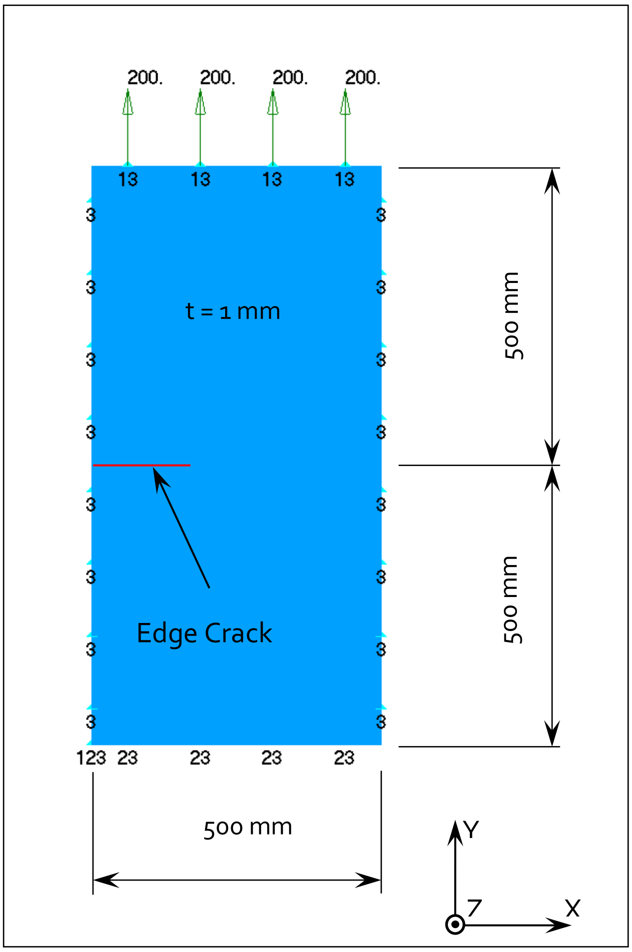


Figure 4-4: Geometry, Loads and Boundary Conditions

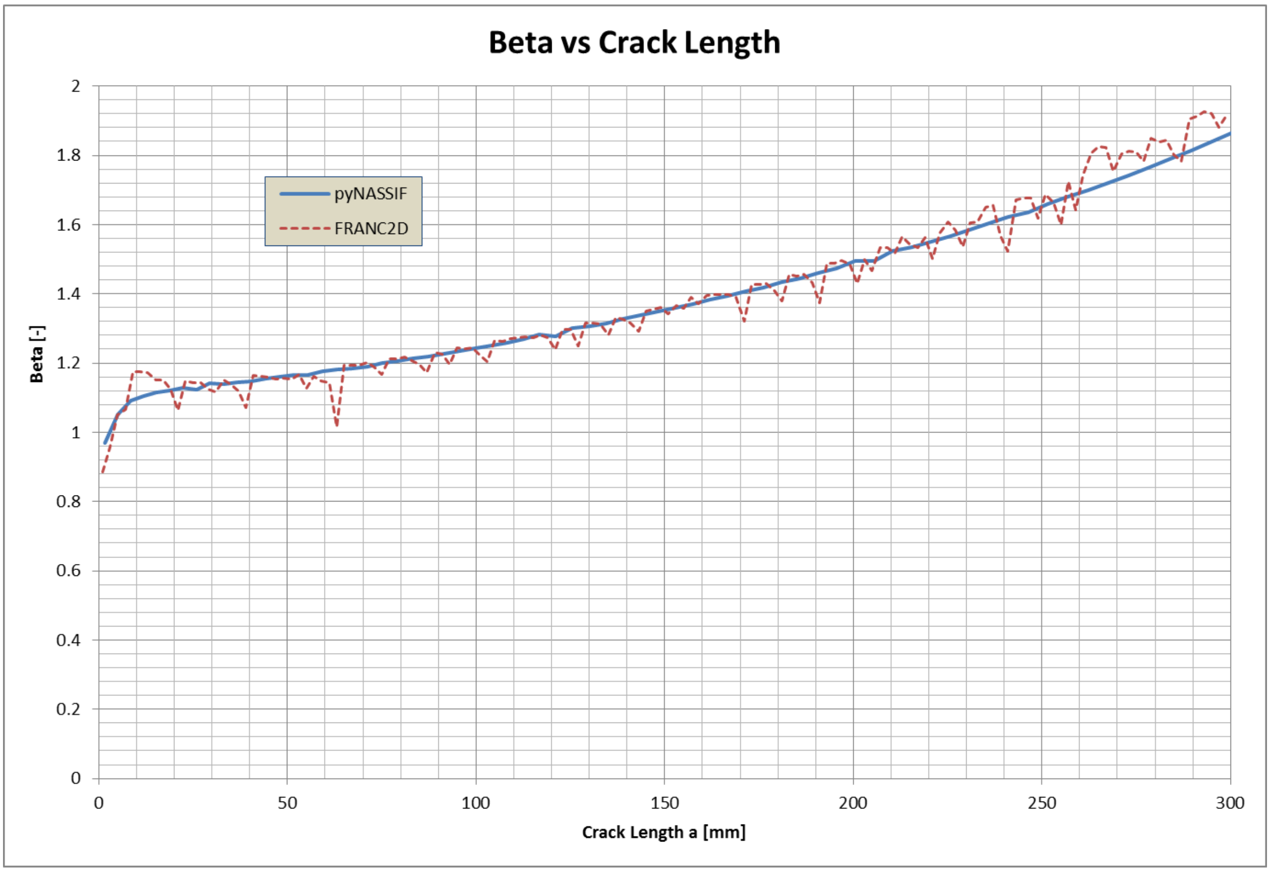


Figure 4-5: β solution for Edge Crack in a Plate under Tension with Guided End, 0 mm < a < 300 mm

4.1.3 Eccentric Hole in a Plate under Tension Loading

Plate is 100mm wide and 1 mm thick and loaded by a constant stress of 100 MPa, see Figure 4-6. Hole diameter is 4 mm with an edge distance of 10mm. As shown in Figure 4-7, β solution calculated by pyNASSIF is in very good agreement with NASGRO and AFGROW results.

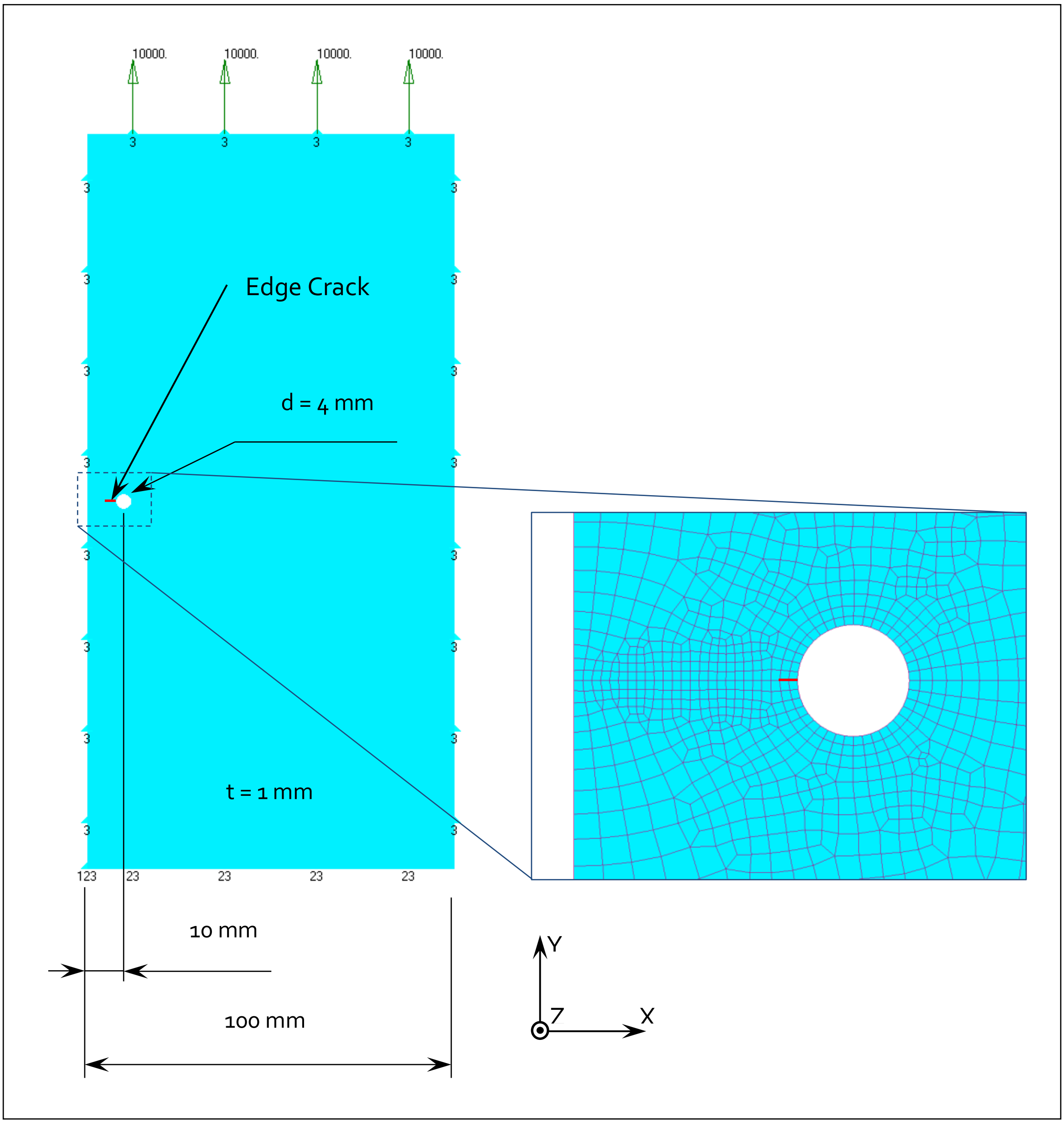


Figure 4-6: Geometry, Loads and Boundary Conditions

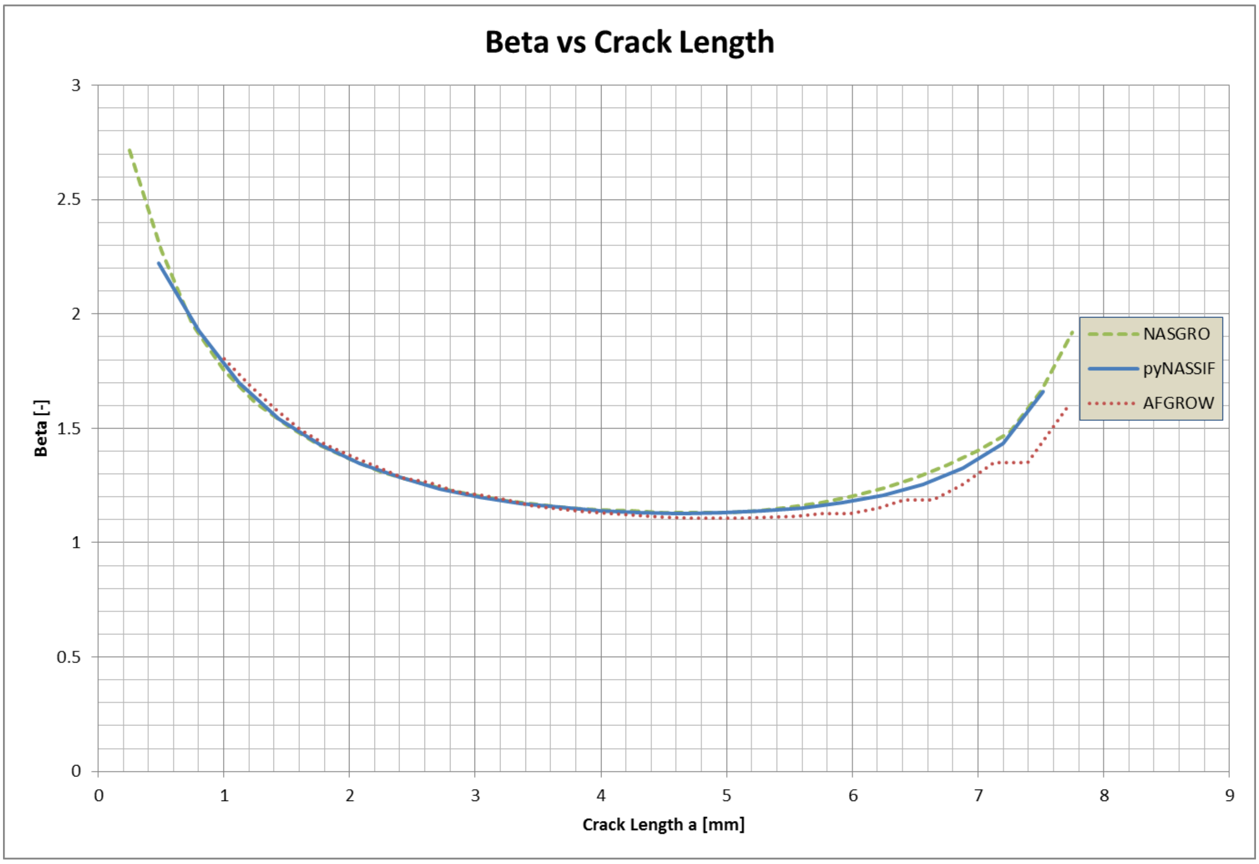


Figure 4-7: β Solution for Edge Crack at Eccentric Hole in Plate under Tension

4.1.3.1 Eccentric Hole in a Plate under Bearing Loading

Plate is 100mm wide and 1 mm thick and hole diameter is 4 mm with an edge distance of 10mm and loaded by a bearing stress of 250 MPa (P = 1000 N, d = 4 mm, t = 1 mm), see Figure 4-8. The bearing load is simulated by a cosine load distribution. As shown in Figure 4-9, β solution calculated by pyNASSIF is in very good agreement with NASGRO and FRANC2D results.

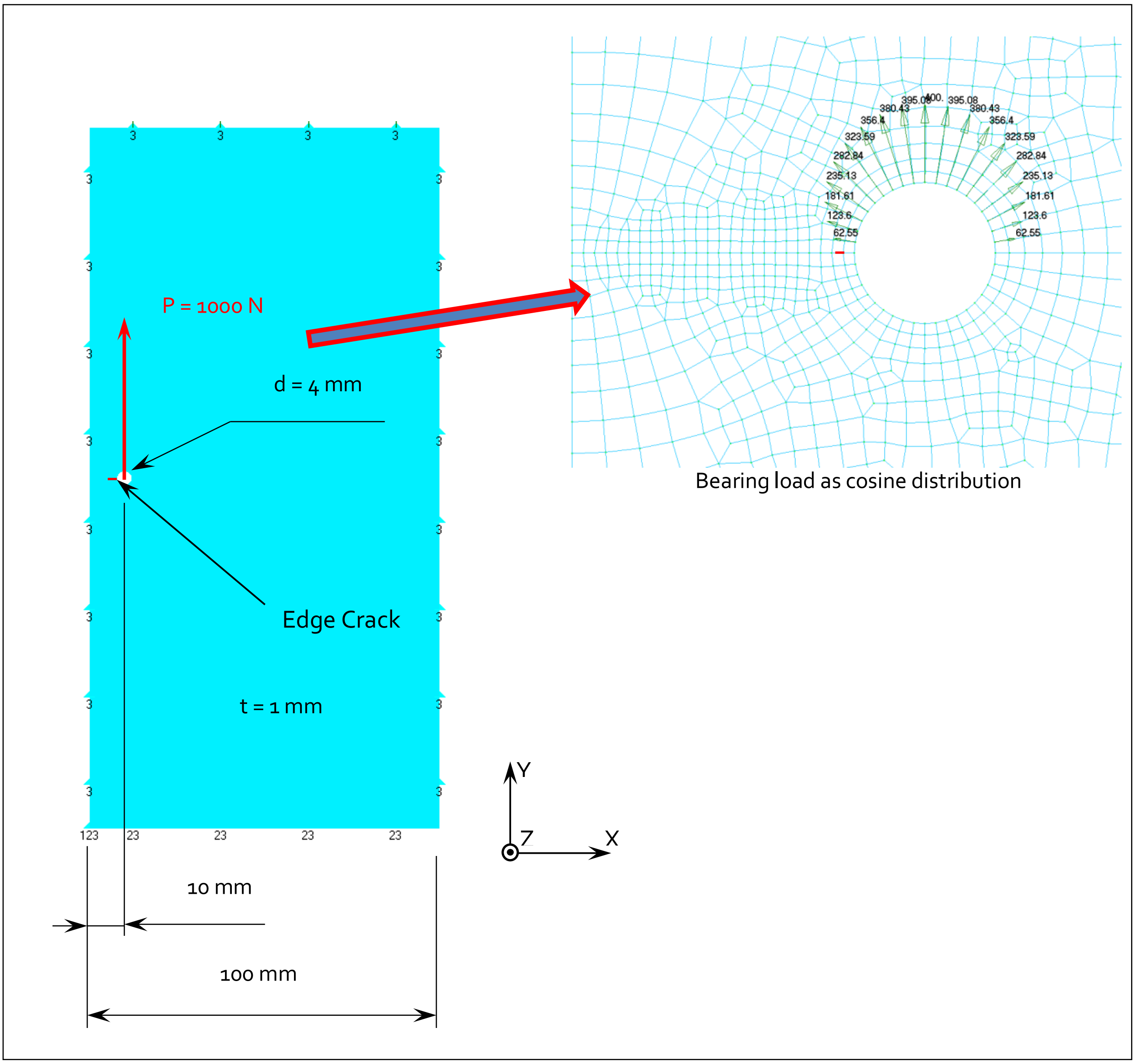


Figure 4-8: Geometry, Loads and Boundary Conditions

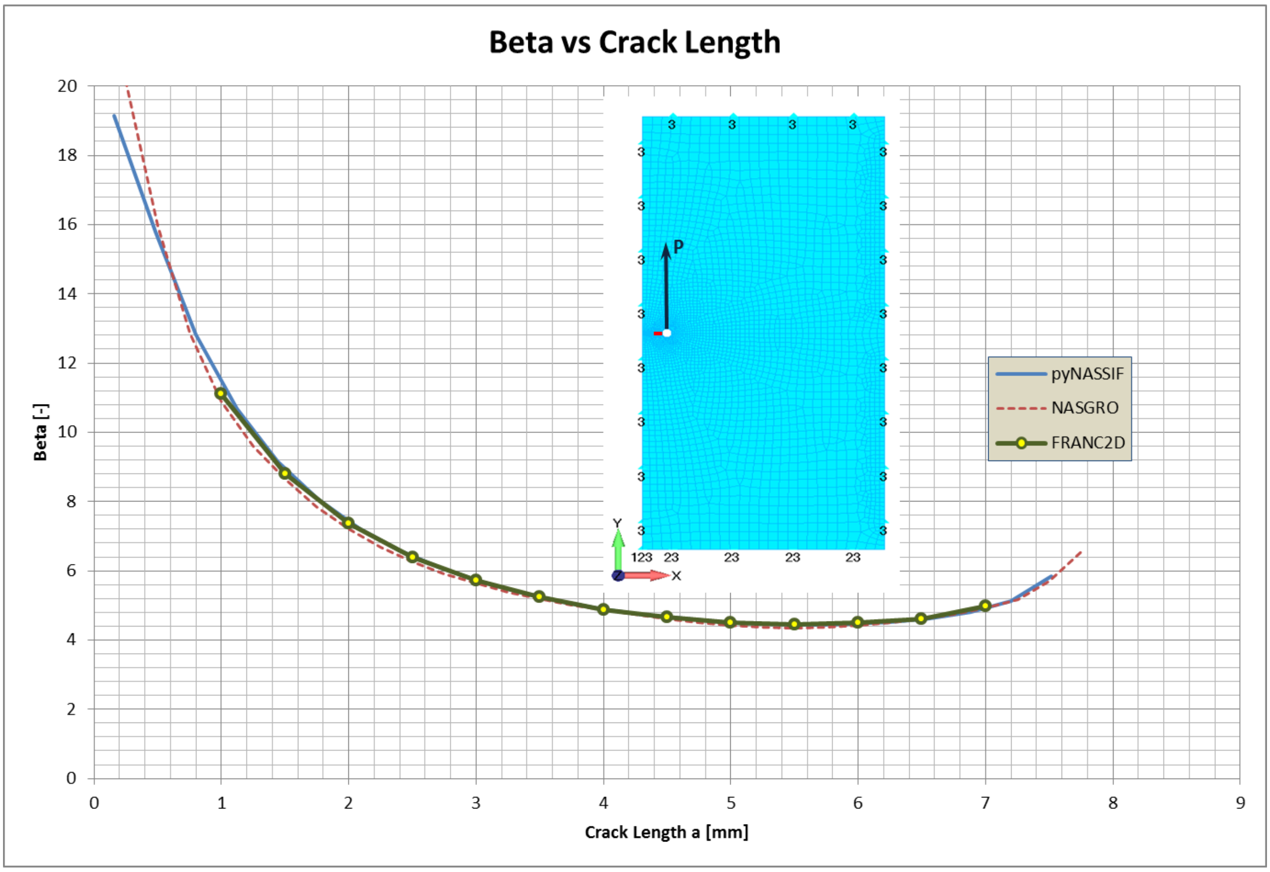


Figure 4-9: β Solution for Edge Crack at Eccentric Hole in Plate under Bearing

4.1.4 Thickness Change

4.1.4.1 From Thin to Thick

Plate is 500 mm wide, 1 mm thick over the first 100, 5 mm thick on the remaining 400 mm and loaded by a tension stress of 100 MPa, see Figure 4-10. As shown in Figure 4-11, β solution calculated by pyNASSIF is in very good agreement with FRANC2D results. It is also shown that the use of the External Work or the Strain Energy leads to the same results.

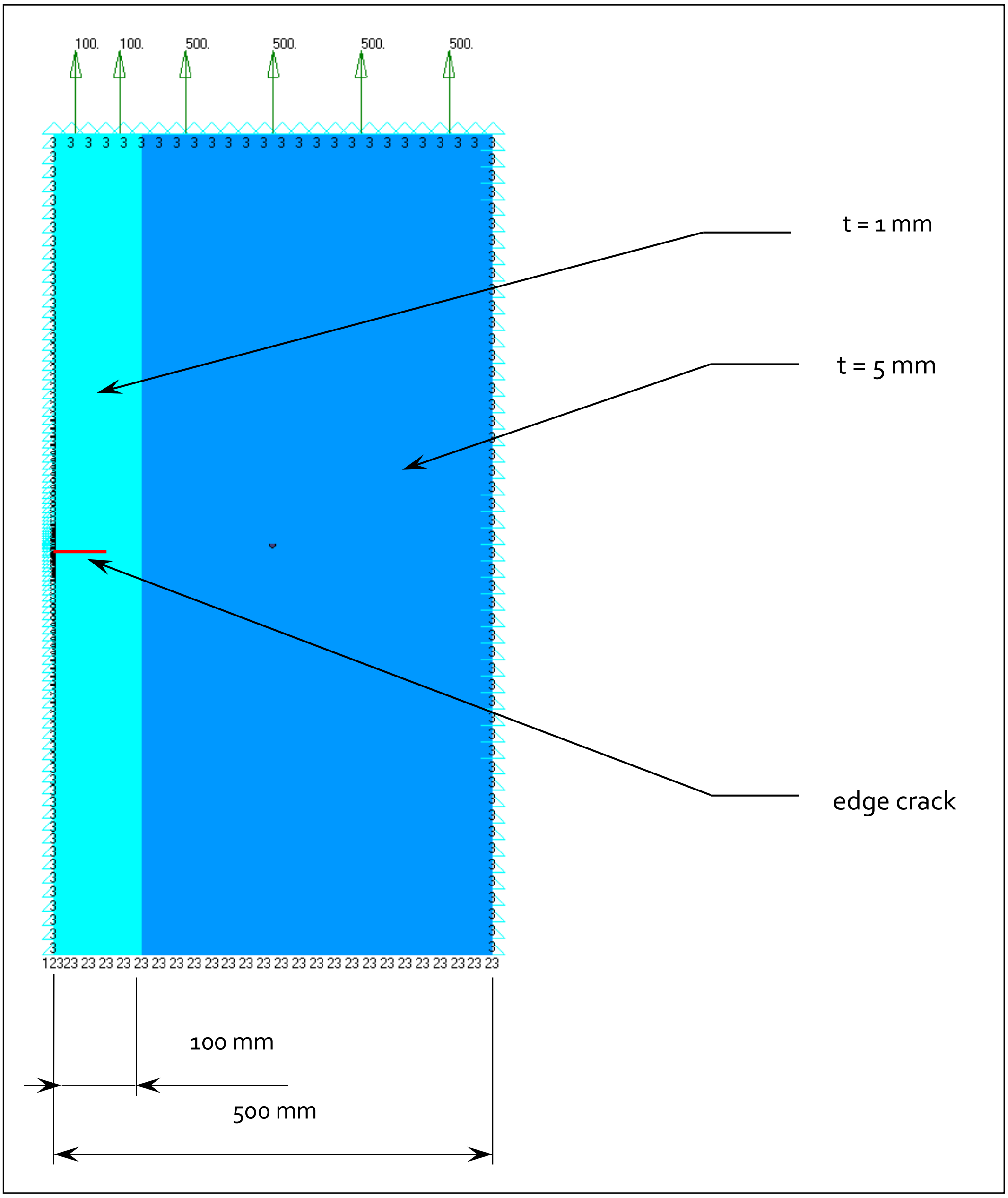


Figure 4-10: Geometry, Loads and Boundary Conditions

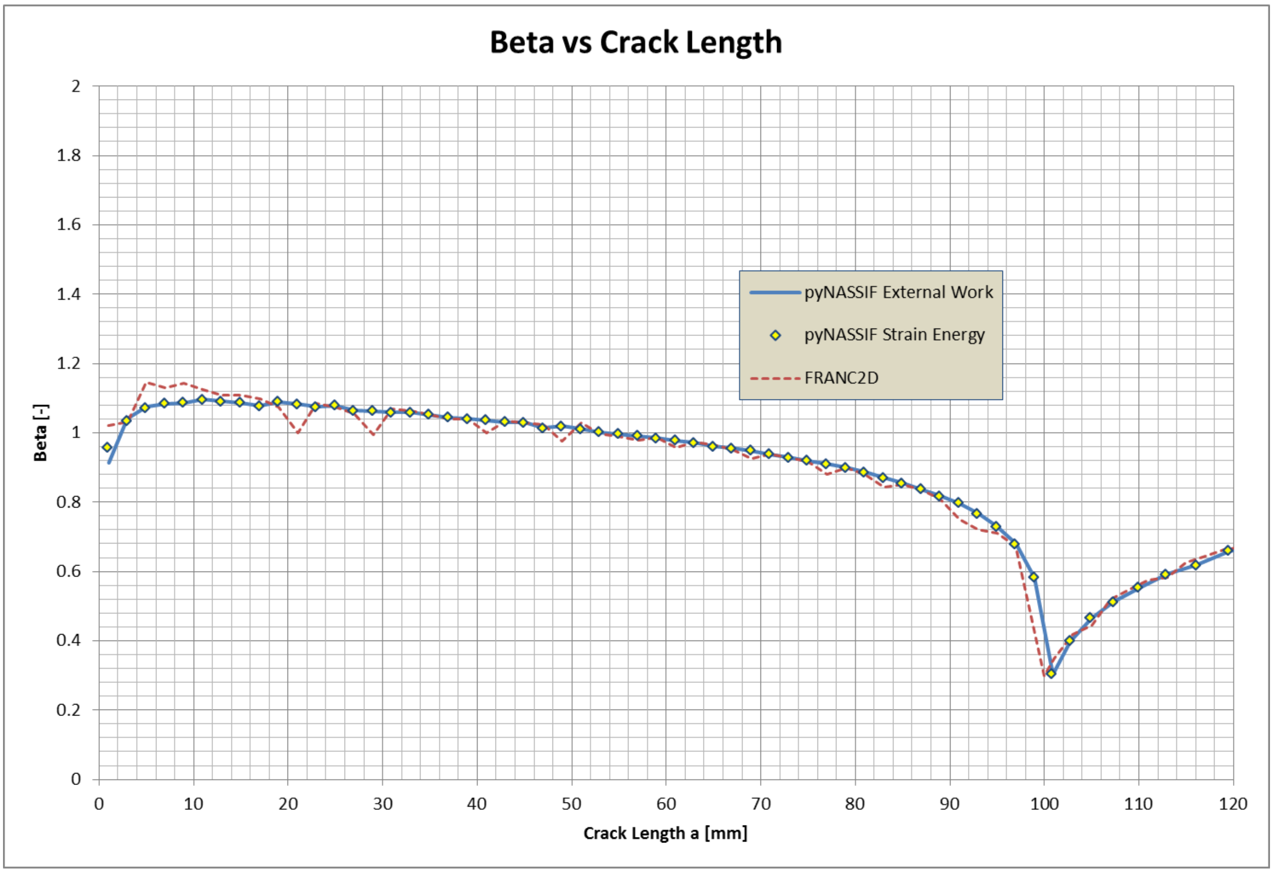


Figure 4-11: β Solution for Edge Crack at a Plate with Thickness Change (Thin to Thick)

4.1.4.1 From Thick to Thin

Plate is 500 mm wide, 5 mm thick over the first 100, 1 mm thick on the remaining 400 mm and loaded by a tension stress of 100 MPa, see Figure 4-12. As shown in Figure 4-13, β solution calculated by pyNASSIF is in very good agreement with FRANC2D results. The difference in the maximum peak at the transition is probably due to difference between the 2 programs regarding the crack path discretization.

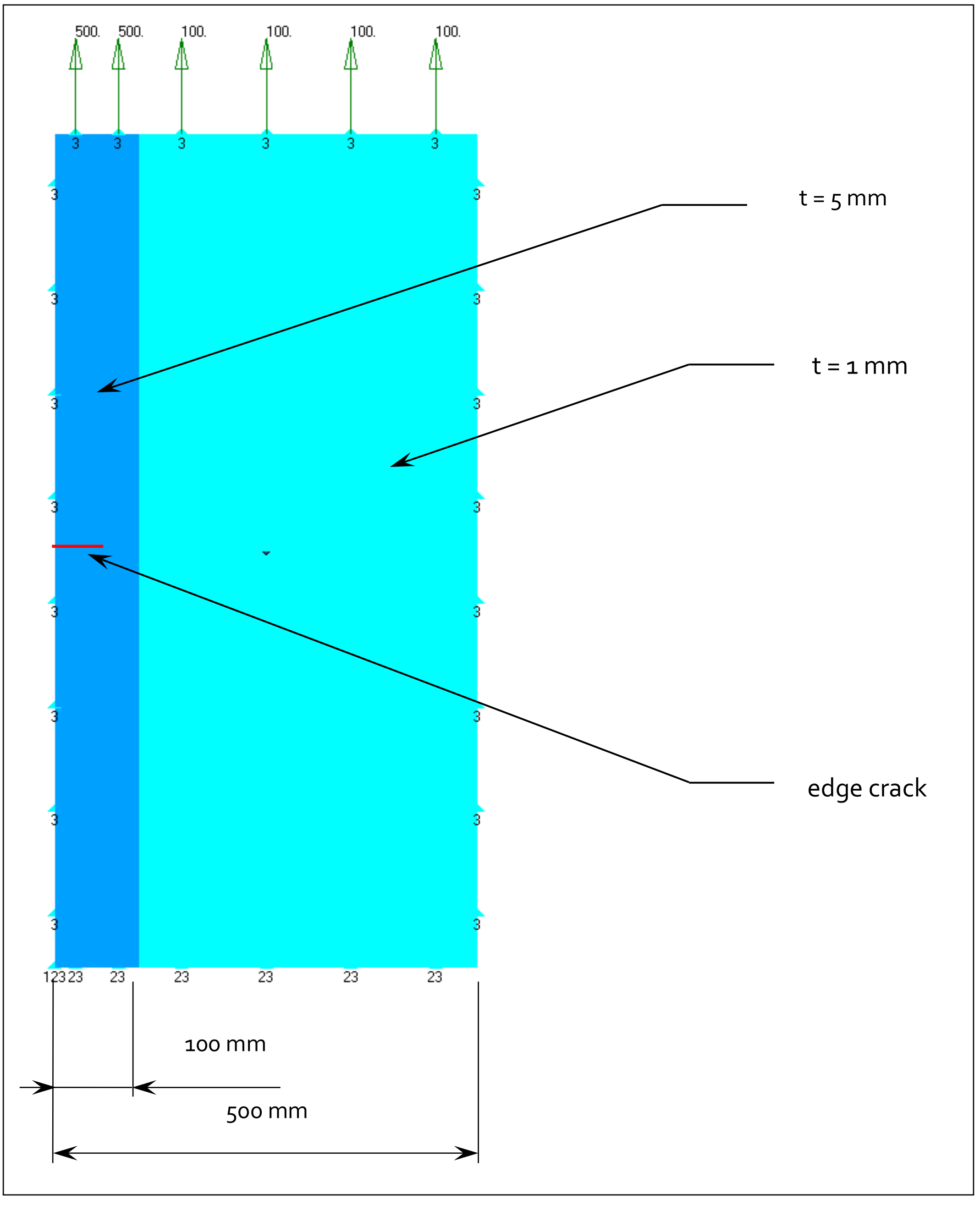


Figure 4-12: Geometry, Loads and Boundary Conditions

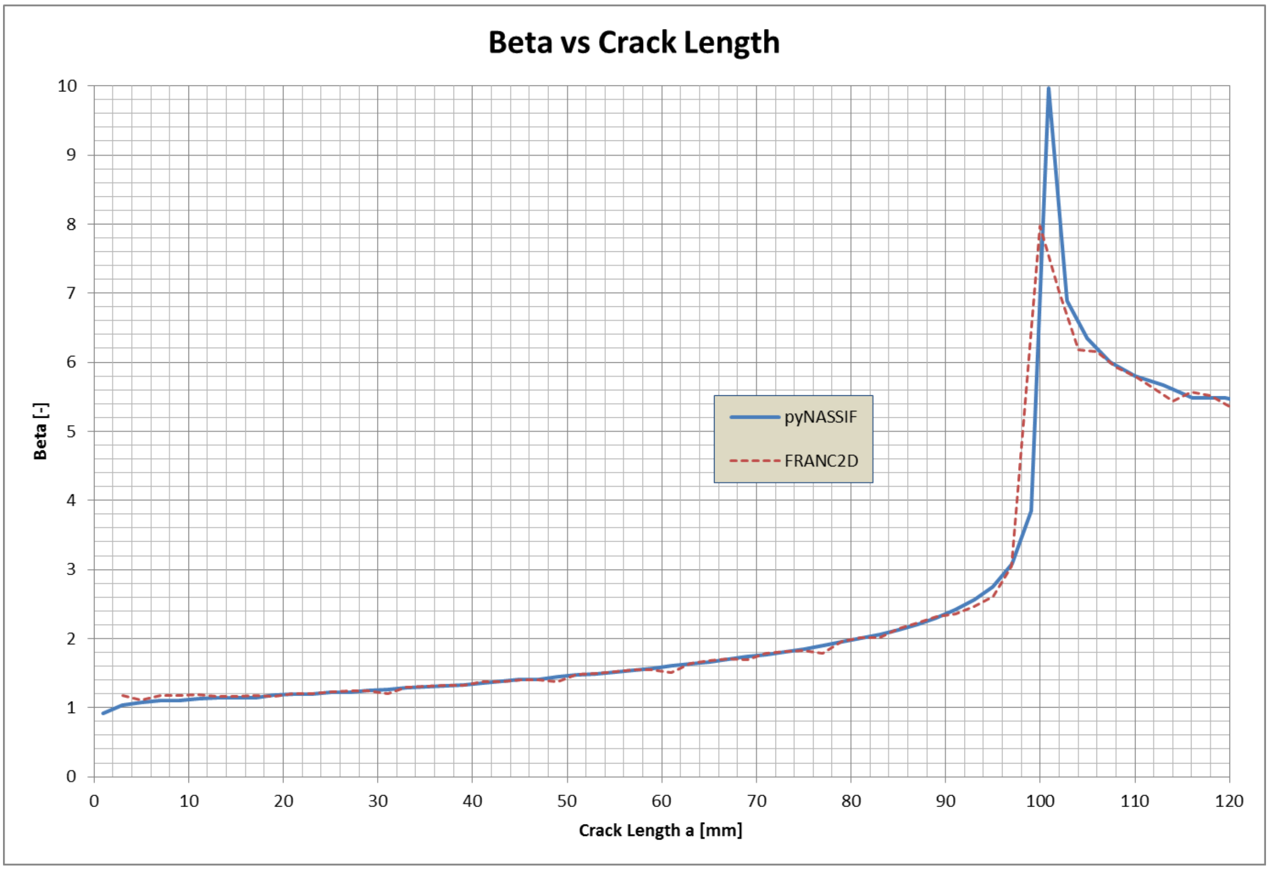


Figure 4-13: β Solution for Edge Crack at a Plate with Thickness Change (Thick to Thin)

4.2 2 TIPS

In this section the solution 122 described in Table 2-1, for which a set of 2 β matrixes (1 matrix per tip) with dependency on Tip 1 and Tip2 lengths is generated, is verified.

4.2.1 Centered Internal Thru Crack in a Plate Loaded in Tension

Plate is 500mm wide and 1 mm thick and loaded by a constant stress of 100 MPa, see Figure 4-14. As shown in Figure 4-15 the β solution calculated by pyNASSIF is in very good agreement with AFGROW [3] and [4]. The impact of the choice of the element shape is highlighted in this figure, where fluctuations in pyNASSIF results are shown at triangular elements nodes. Therefore, in order to minimize results fluctuations it is recommended to use exclusively quadrilateral elements along the crack path. If this is not possible, polynomial fit of the pyNASSIF results could be used to smooth the data.

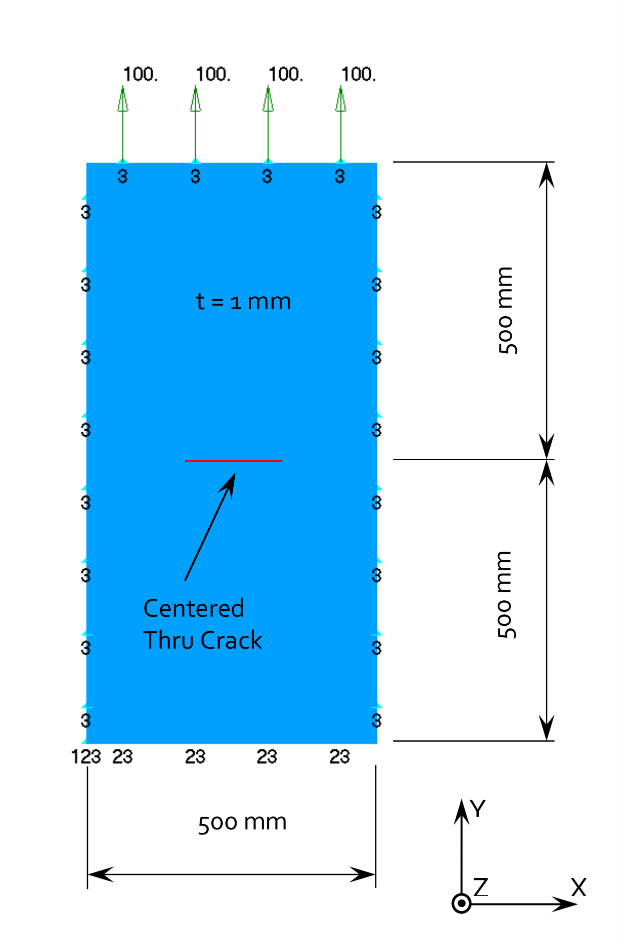


Figure 4-14: Geometry, Loads and Boundary Conditions

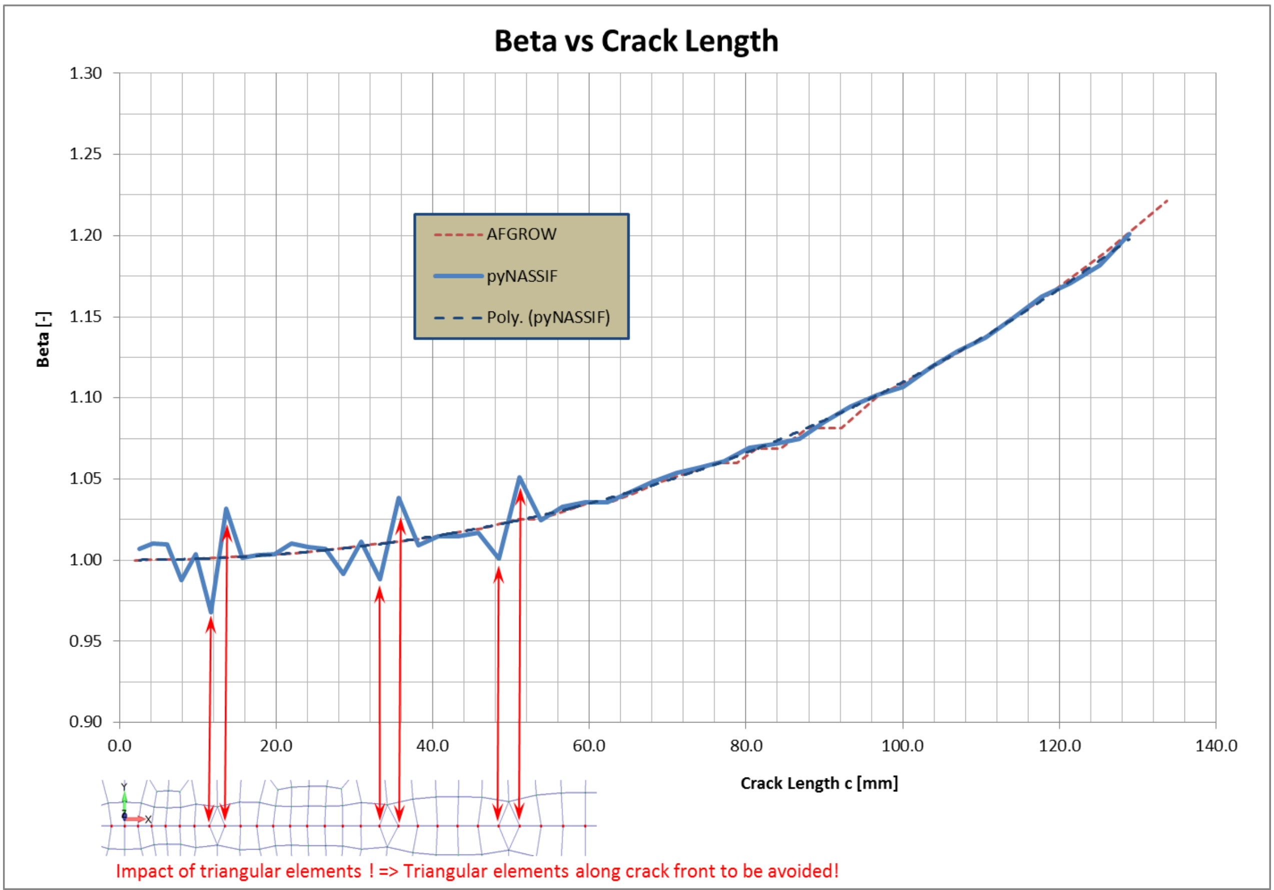


Figure 4-15: β Solution for Centered Internal Thru Crack in a Plate Loaded in Tension

4.2.1 Eccentered Internal Thru Crack in a Plate Loaded in Tension

Plate is 500mm wide and 1 mm thick with a thru crack positioned at 125 mm from edge and loaded by a constant stress of 100 MPa, see Figure 4-16. The diagonal terms of both β matrixes were extracted, which means that same crack increment was applied on both tips and compared to AFGROW runs using same crack pattern. As shown in Figure 4-17 the β solutions calculated by pyNASSIF are in very good agreement with AFGROW [3] and [4] for both tips.

In order to check AFGROW capacity to deal with 2 β matrixes CG analysis was performed using AFGROW model 1000 (User-Defined Part Through Crack – Standard Solution). Crack length ratio a/c was set to 1 to enforce same crack increment on both Tips. As shown in Figure 4-18 the β values interpolated by AFGROW are very close to pyNASSIF values. Therefore, AFGROW model 1000 (User-Defined Part Through Crack – Standard Solution) is found to be a good solution to perform the matrices interpolation.

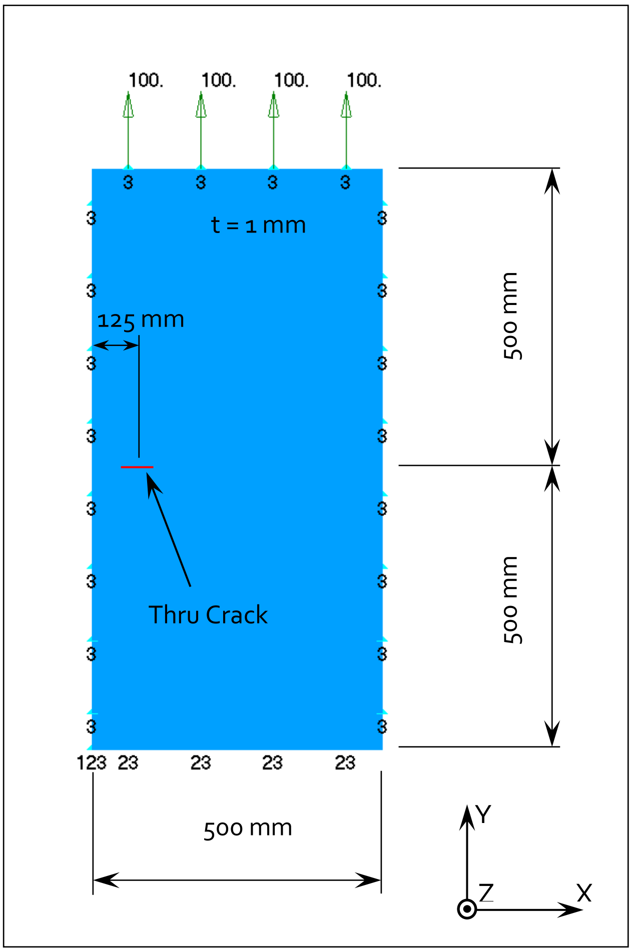


Figure 4-16: Geometry, Loads and Boundary Conditions

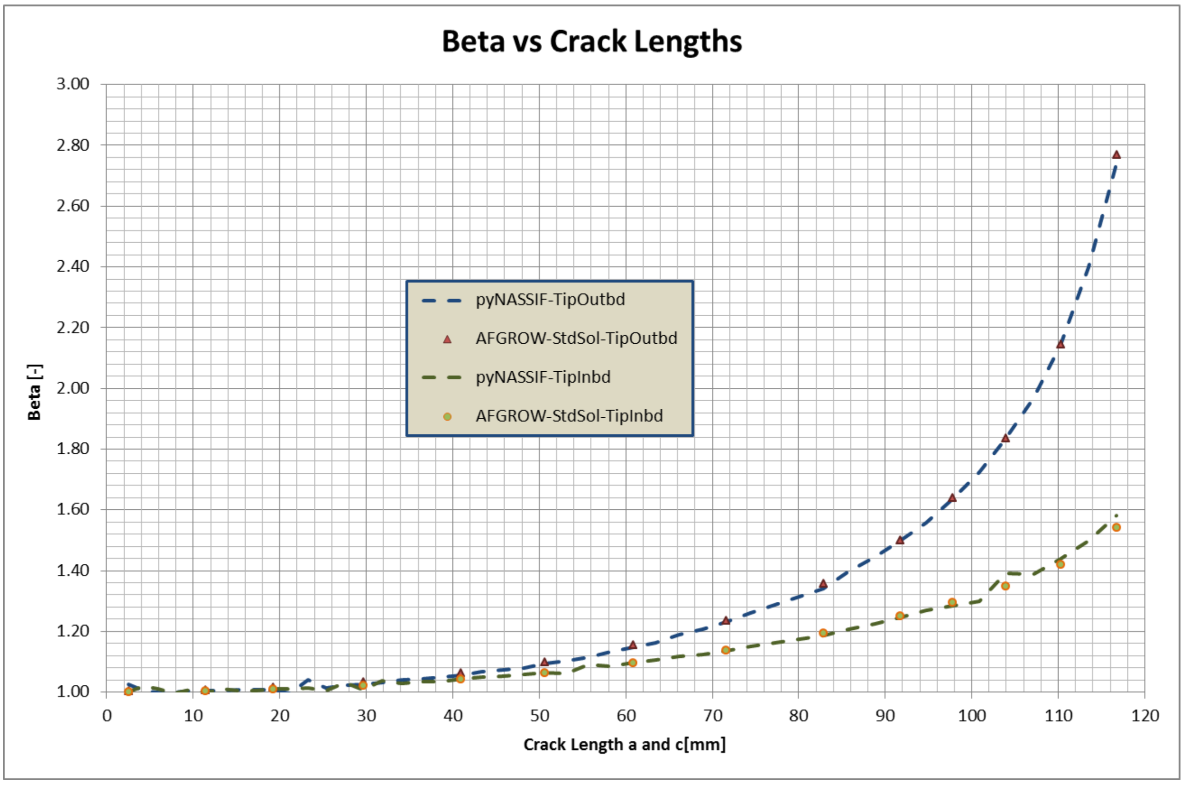


Figure 4-17: β Solutions for Eccentered Internal Thru Crack in a Plate Loaded in Tension

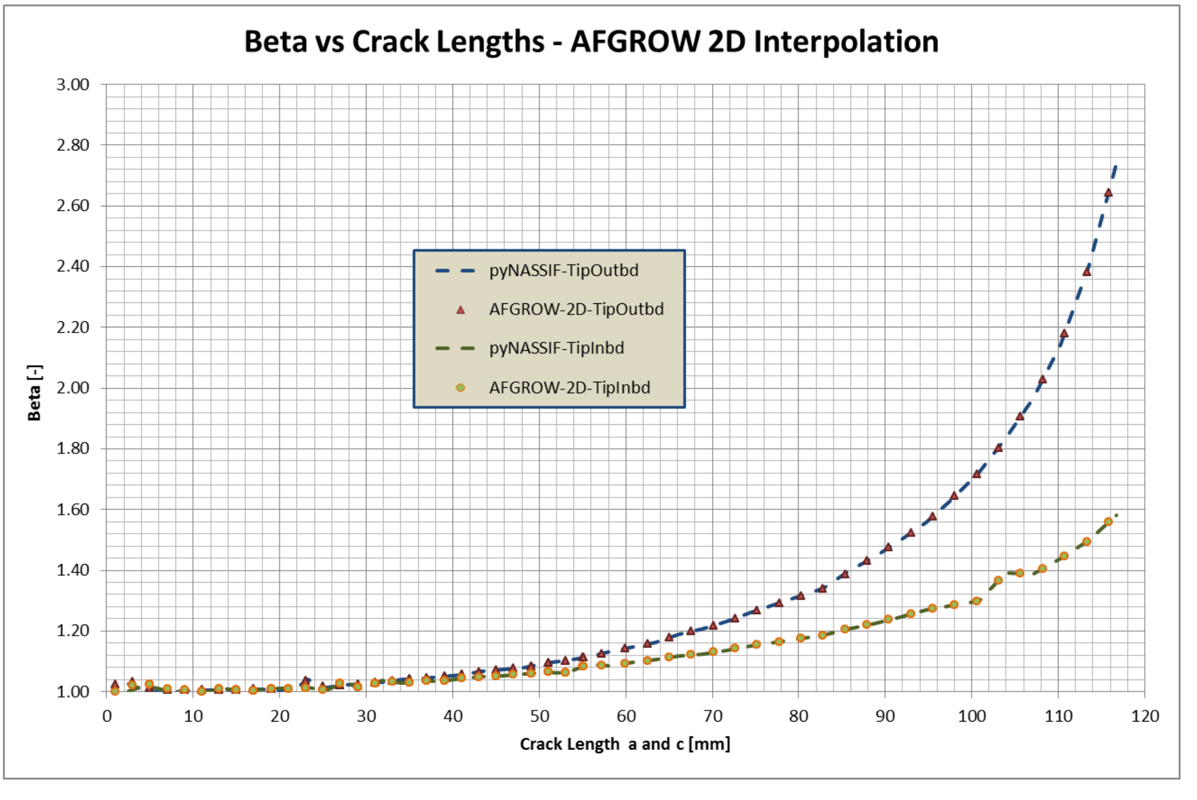


Figure 4-18: Comparison between pyNASSIF and AFGROW Interpolated β Values

5 LIMITATIONS AND RECOMMENDATIONS

5.1 CRACK PATH

pyNASSIF needs the crack path to be defined in advance by the analyst (no automatic crack path calculation). Depending on the crack propagation mode (mode I is tension opening, mode II is in-plane shear and mode III is out-of-plane shear, see Figure 5-1) the criteria for crack path definition is different. For mode I the crack tends to grow in a plane perpendicular to the maximum principal stress while for modes II and III the crack tends to grow in the direction of the maximum shear.

In most cases, a crack path perpendicular to the maximum principal stress field is a good approximation. However, it the responsibility of the analyst to ensure that the crack path defined in the NASTRAN model is realistic. In case of doubt, a possible solution is to use pyNASSIF along both potential crack paths and select the one leading to the highest K or β for subsequent analyses.

A typical example of a simplified crack path search analysis is presented Figure 5-2 and Figure 5-3.

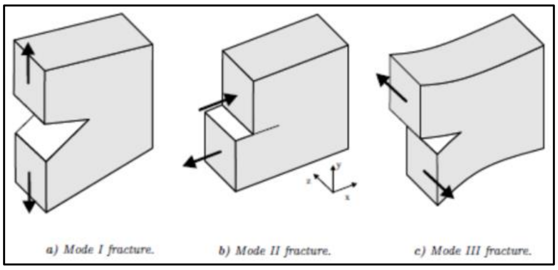


Figure 5-1: Modes of Fracture (I, II & III)

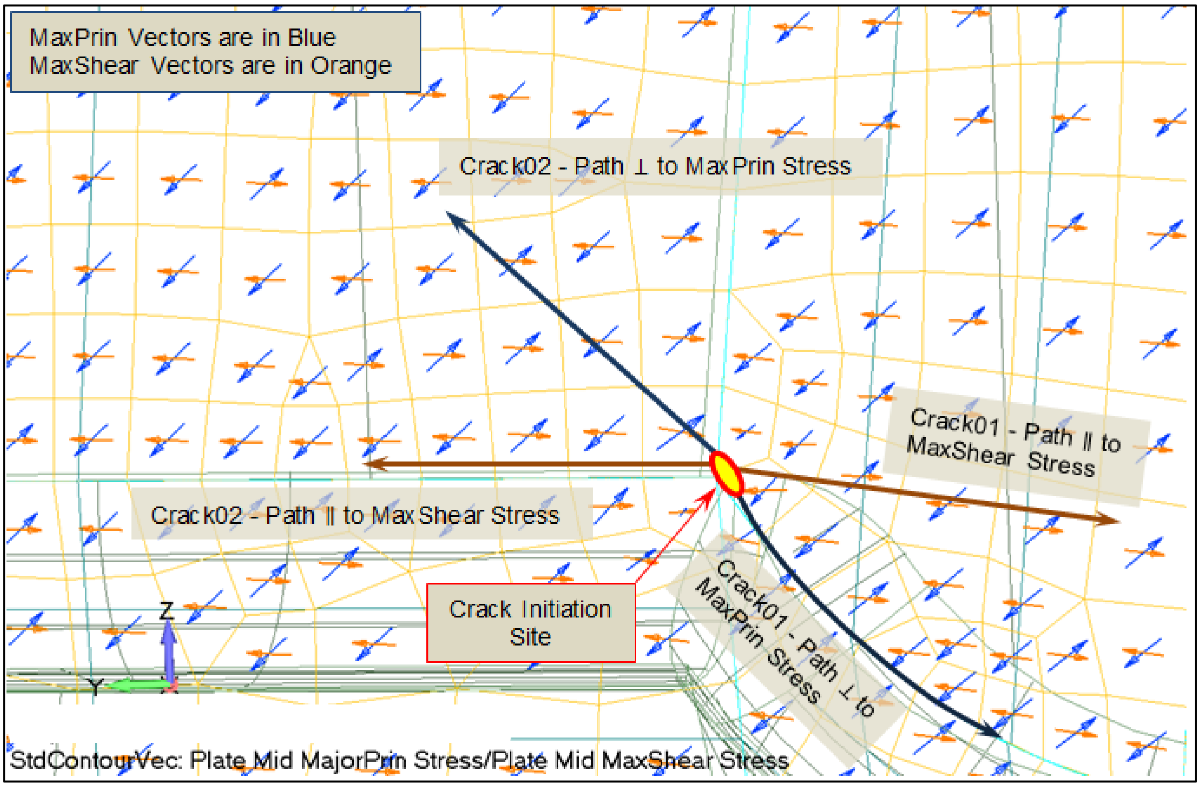
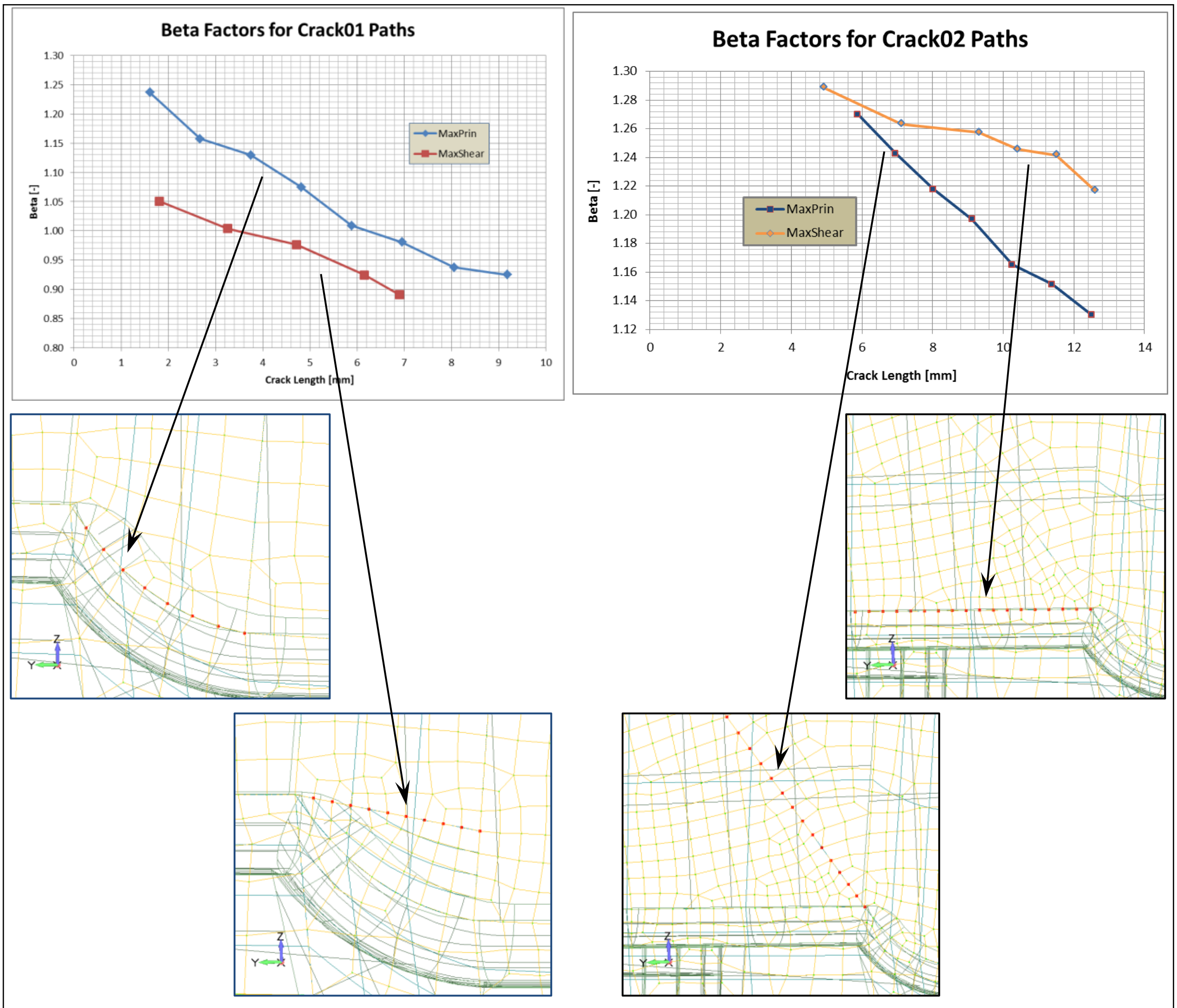


Figure 5-2: Typical Selected Crack Paths



⇨ MaxPrin path is selected on Crack01 tip because it leads to higher β solution

⇨ MaxShear path is selected on Crack02 tip because it leads to higher β solution

Figure 5-3: Selection of Path leading to Maximum β

5.2 Plain Stress / Plain Strain

There is no automatic detection of the plain stress / plain strain condition implemented in pyNASSIF and in this software version it is assumed that plain stress conditions apply. Therefore, it is up to the analyst to check, which of the plain stress / plain strain condition applies and to correct the pyNASSIF outputs accordingly.

5.3 Influence of crack Discretization

More detailed studies have to be performed in order to assess the impact of the crack increment size on the results. However, the following guidelines are provided based on experience with pyNASSIF:

1. When initial crack size is zero, start with crack increments da $\leq$ 0.5 x t
2. For small crack length (a < 10 x t) use crack increments da ≤ t
3. For intermediate crack length (10 x t < a < 20 x t) use crack increments da $\cong$ t
4. For longer crack length (a > 20 x t) use crack increment no longer than da = a / 10 or no longer than da = 5 x t

The guidelines given above can lead to External Work or Strain Energy difference between two crack increments, which is too low and therefore not captured by pyNASSIF (rounding error). In this case the model is too big and a sub-model using free-body loads or free-body displacements is required to reduce the External Work.

5.4 Influence of Element Shape

Experience with pyNASSIF has shown that when a mix of quadrilateral and triangular elements are present along the crack line fluctuations in the results will occur. Therefore, it is recommended to use exclusively quadrilateral elements along the crack path. If this is not possible, polynomial fit of the pyNASSIF results could be used to smooth the data. Use of mesh with only triangular elements has not been validated yet.

5.5 Solution 122 Mesh

For best results the size of the elements should be similar between same increments on Tip 1 and Tip 2.

5.6 Results for first increment

Usually the SIF for the first increment is too low. This is a consequence of the discretization process coupled with the definition of the crack length and crack increment used in pyNASSIF. Therefore, it is recommended to ignore the first K and β output line for solutions 110, 120 and 121.

Note that solution 122 already accounts for this effect by removing the first line of the β matrices.

6 References

[1] AFWAL-TR-82-3073, USAF Damage Tolerant Design Handbook, May 1984

[2] Compendium of Stress Intensity Factors, Rooke & Cartwright, 1976

[3] AFGROW, Version 5.1.6.17, 05 January 2011

[4] AFGROW, Version 5.2.1.18, 17 February 2014

[5] NASGRO, Version 3.0, December 2000

[6] FRANC2DL, Version 2.3, 15 January 2010